

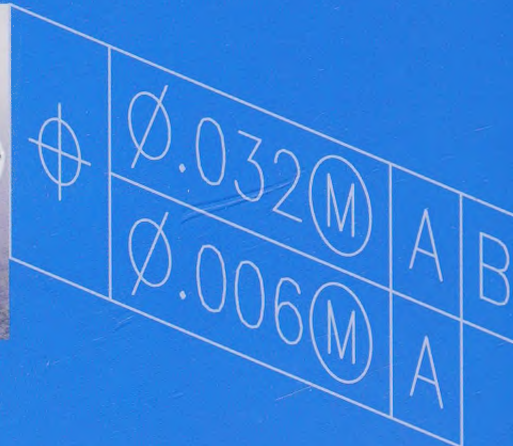
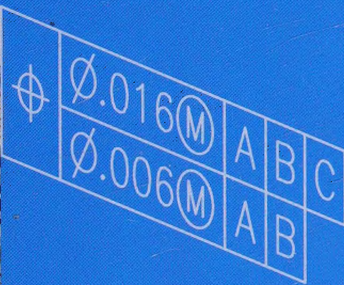
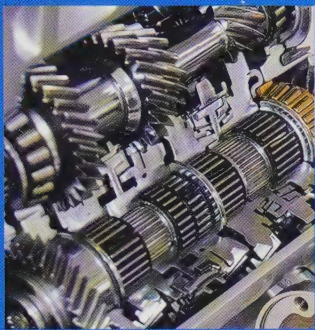
GD&T

Application and Interpretation

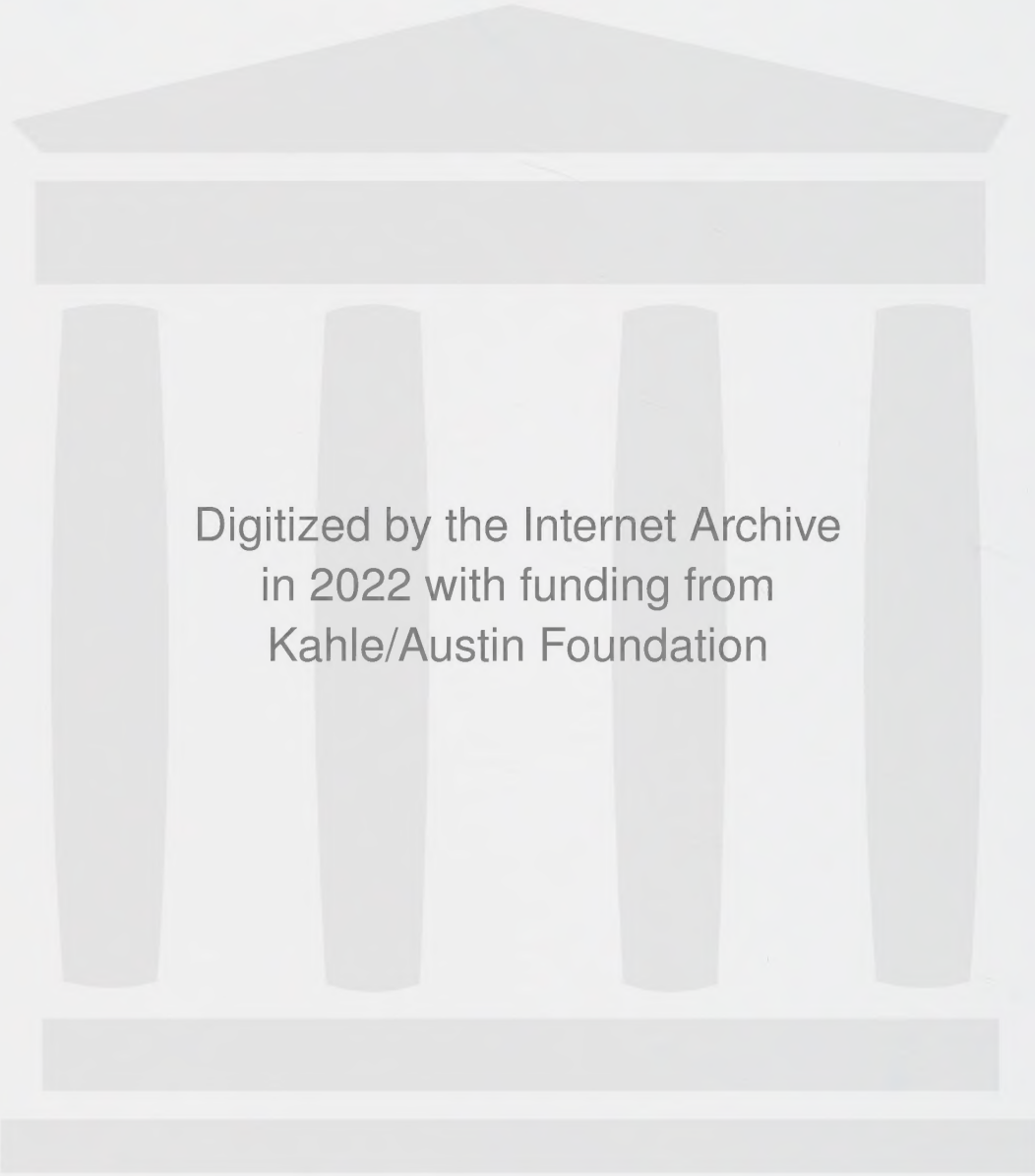
Sixth Edition

Bruce A. Wilson

The author is GDTP certified by ASME in accordance with the qualifications of ASME 14.5.2 in the Senior level.



Based on the ASME Y14.5-2009 standard



Digitized by the Internet Archive
in 2022 with funding from
Kahle/Austin Foundation

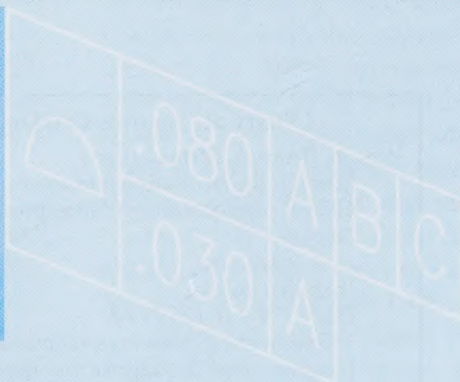
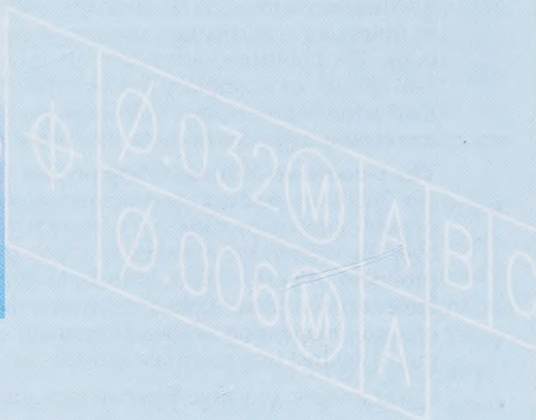
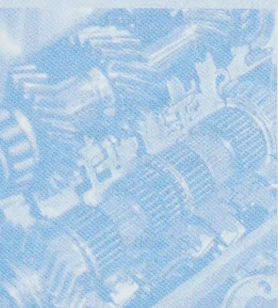
GD&T

Application and Interpretation

Sixth Edition

by

Bruce A. Wilson



Publisher

The Goodheart-Willcox Company, Inc.

Tinley Park, IL

www.g-w.com

Copyright © 2016

by

The Goodheart-Willcox Company, Inc.

Previous editions copyright 2010, 2005, 2001, 1996, 1992

All rights reserved. No part of this work may be reproduced, stored, or transmitted in any form or by any electronic or mechanical means, including information storage and retrieval systems, without the prior written permission of The Goodheart-Willcox Company, Inc.

Manufactured in the United States of America.

Library of Congress Catalog Card Number 2014037225

ISBN 978-1-63126-113-8

3 4 5 6 7 8 9 – 16 – 19 18 17

The Goodheart-Willcox Company, Inc. Brand Disclaimer: Brand names, company names, and illustrations for products and services included in this text are provided for educational purposes only and do not represent or imply endorsement or recommendation by the author or the publisher.

The Goodheart-Willcox Company, Inc. Safety Notice: The reader is expressly advised to carefully read, understand, and apply all safety precautions and warnings described in this book or that might also be indicated in undertaking the activities and exercises described herein to minimize risk of personal injury or injury to others. Common sense and good judgment should also be exercised and applied to help avoid all potential hazards. The reader should always refer to the appropriate manufacturer's technical information, directions, and recommendations; then proceed with care to follow specific equipment operating instructions. The reader should understand these notices and cautions are not exhaustive.

The publisher makes no warranty or representation whatsoever, either expressed or implied, including but not limited to equipment, procedures, and applications described or referred to herein, their quality, performance, merchantability, or fitness for a particular purpose. The publisher assumes no responsibility for any changes, errors, or omissions in this book. The publisher specifically disclaims any liability whatsoever, including any direct, indirect, incidental, consequential, special, or exemplary damages resulting, in whole or in part, from the reader's use or reliance upon the information, instructions, procedures, warnings, cautions, applications, or other matter contained in this book. The publisher assumes no responsibility for the activities of the reader.

The Goodheart-Willcox Company, Inc. Internet Disclaimer: The Internet resources and listings in this Goodheart-Willcox Publisher product are provided solely as a convenience to you. These resources and listings were reviewed at the time of publication to provide you with accurate, safe, and appropriate information. Goodheart-Willcox Publisher has no control over the referenced websites and, due to the dynamic nature of the Internet, is not responsible or liable for the content, products, or performance of links to other websites or resources. Goodheart-Willcox Publisher makes no representation, either expressed or implied, regarding the content of these websites, and such references do not constitute an endorsement or recommendation of the information or content presented. It is your responsibility to take all protective measures to guard against inappropriate content, viruses, or other destructive elements.

Cover images: yuyangc/Shutterstock.com, tratong/Shutterstock.com, Chirasak Tolertmongkol/Shutterstock.com

Library of Congress Cataloging-in-Publication Data

Wilson, Bruce A. (Bruce Allen)

[Design dimensioning and tolerancing]

GD&T : application and interpretation / Bruce A. Wilson. -- 6th edition.

pages cm

Includes index.

ISBN 978-1-63126-113-8

1. Engineering drawings--Dimensioning. 2. Tolerance (Engineering). 3. Engineering design. I. Title. II. Title: G.D.&T. III. Title: GD&T. IV. Title: GD and T. V. Title: General dimensioning and tolerancing.

T357.W47 2016

604.2'43--dc23

2014037225

Preface

A majority of commercial and military industries require their engineers, designers, and anyone involved in creating design documentation to be knowledgeable in proper dimensioning and tolerancing methods. This knowledge is important in product design, production equipment design, and tooling design. People that work in the procurement and manufacturing of parts are also expected to understand dimensions and tolerances. Machinists, machine programming personnel, tool makers, inspectors, and manufacturing engineers must understand dimensions and tolerances since they work to the specifications created by the engineers and designers.

The guidelines for consistent and clear application of dimensions and tolerances are defined by national standards as written by the American Society of Mechanical Engineers (ASME). The number of companies requiring compliance with national standards is continually growing. Standards written by the International Organization for Standardization (ISO) also define dimensioning and tolerancing requirements and are becoming widely used. The ISO methods are similar to ASME standards, but there are detailed differences.

Proper application of dimensions and tolerances is an important part of providing complete documentation of product requirements. ASME Y14.5 is the authoritative document for defining dimensioning and tolerancing symbols and application methods. ASME Y14.41 provides further explanation specifically applicable to digital product definition (computer-aided design). *GD&T: Application and Interpretation* provides an expanded explanation of the material contained in these and other applicable ASME standards. Additional standards that affect calculation and application of tolerances are explained and referenced within this text when appropriate.

The subject of this text ranges from the fundamentals of dimension application to extended principles of tolerance application. It provides introductory information on GD&T for an entry-level class while also providing the information needed for in-depth understanding and use as a reference book in education and industry. Although the importance of dimensioning fundamentals is reduced by the utilization of solid models to establish dimensional requirements, this

text provides an explanation of the fundamentals since some companies still create fully dimensioned product definition. Not everyone will need to know the dimensioning fundamentals since industry is transitioning away from the visible display of dimensions and allowing the computer-generated geometry to define dimension values.

Tolerance application and interpretation explanations in this textbook are included for all of the categories of tolerances in ASME Y14.5. Explanations are primarily based on the current version of the standard, ASME Y14.5-2009. Past practices as defined in earlier releases of ASME Y14.5 are briefly explained to provide information for people who encounter tolerance specifications that were created prior to issuance of the current standard. Past practices, where included in this text, are clearly distinguished from current practices.

GD&T: Application and Interpretation contains twelve chapters. A list of major topics is provided at the beginning of the chapter. A chapter summary is given at the end. Chapter review materials include questions and application problems. Additional problems are available in the Study Guide for *GD&T: Application and Interpretation* and in the ExamView® Assessment Suite, which contains more than 1700 questions.

Generally, tolerancing terms are defined within the text when they are first introduced. Their definitions are repeated in the Glossary to permit easy future reference. Other technical terms are italicized when they first appear. They may be defined within the text or in the glossary.

Dimensioning and tolerancing terms used within this textbook generally conform to those used in the ASME standards. The terminology is very specific to make possible an accurate explanation of tolerance application and interpretation. After a detailed term is defined, there are instances where further application of the term is simplified. An explanation may refer to the centerline or “axis” of a mating hole, rather than use a complex reference to the “axis of the unrelated actual mating envelope”.

Illustrations are used extensively to clarify explanations. Each figure generally introduces only one new concept. Figures are not complicated with detail that is unrelated to the concept being explained. This makes it possible for the reader to quickly see the necessary information.

Color is used to separate explanation data from the main portion of the figures. Color is also used to highlight instructional information such as tolerance placement requirements and tolerance zone boundaries.

Interpretation figures are provided to show the tolerance zones resulting from tolerance specifications. Tolerance zones and part variations in these figures are sufficiently exaggerated to make them visible.

Uppercase letters are used for notations that are part of the drawing. Lowercase letters are used for instructional notes and information. One exception is that degrees of freedom notations, within a drawing, are made with lowercase letters as required by the ASME standards.

Dimensioning and tolerancing of a part requires application of principles from the ASME Y14.5 standard. When defining tolerances in digitally defined products (using CAD), conformance with ASME Y14.41 is desirable. *GD&T: Application and Interpretation* shows how to apply the principles in these standards to many situations and offers suggestions regarding the extension of the principles to situations not shown in the standards. Care must be taken in the extension of principles. Otherwise, a violation of the standards or unclear requirements could result. Compliance with the standards is ultimately the responsibility of the person applying or interpreting dimensions and tolerances.

About the Author

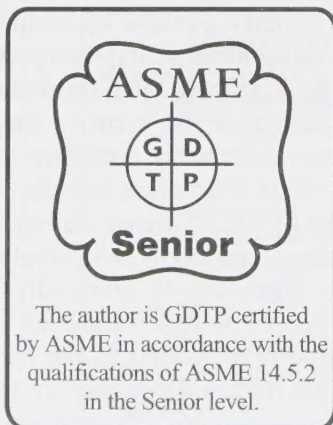
Bruce A. Wilson, author and illustrator of *GD&T: Application and Interpretation*, has an international reputation as a leader in Dimensional Management and Geometric Dimensioning and Tolerancing (GD&T). He is certified by the American Society of Mechanical Engineers (ASME) as a Senior Level Geometric Dimensioning and

Tolerancing Professional. He has more than 25 years participation in ASME subcommittee Y14.5 standards development. He served as the leader of the positional tolerances section of the standard for more than 14 years and now is chairman of the Y14.5 subcommittee.

Mr. Wilson has extensive training and consulting experience. He has industrial and academic experiences that provide the expertise and practical knowledge necessary to write a dimensioning and tolerancing text that is technically correct and applicable to today's industrial needs. Mr. Wilson has taught GD&T in Europe and the United States. His experience includes curriculum development, teaching, management of training programs, and instructor development. Areas of concentration include Dimensioning and Tolerancing for engineering and manufacturing, Computer-Aided Design, Dimensional Management, and New Designer Training. Mr. Wilson led training programs for thousands of design engineers, manufacturing engineers, machinists, tool designers, tool makers, quality engineers, inspectors, supplier managers, and managers. He has advised executives regarding the technical needs and health of their organizations.

Mr. Wilson's industrial experience includes responsibilities ranging from design engineer to program manager and industrial consultant. He was recognized as a Technical Fellow in The Boeing Company. His design engineering experience includes precision mechanisms, electromechanical, optics packaging, electronics packaging, pneumatics, and aircraft structures. His experience has taken place in companies ranging from 20 to 170,000 employees, producing everything from bicycle wheels to spacecraft.

Mr. Wilson's extensive design experience has provided a thorough exposure to dimensioning and tolerancing requirements as they apply to various applications. He is an industrial leader in the discipline of dimensional management. He led teams to establish product system requirements, complete 3D variation analysis, apply tolerances to detail parts and assemblies, communicate requirements to production, define locating methods for major assemblies, and implement quality assurance measures related to dimensional variation. His variation analyses and assembly concepts have resulted in product cost savings greater than 50% and elimination of assembly tooling costs for major aircraft structure. The processes he explored are now implemented widely. Mr. Wilson has multiple United States patents for calculation methods used to assess variation effects.



Mr. Wilson has contributed to the development of national and international standards since 1986 through his participation and membership on the ASME Y14.5 subcommittee. He has served as a member of the ASME Y14, Y14.1, Y14.2, Y14.3, and Y14.5 subcommittees and chaired both the Y14.3 and Y14.5 subcommittees. He has participated in United States technical advisory groups for ISO dimensioning standards.

Mr. Wilson's education includes Bachelor of Science and Master of Science degrees from Southern Illinois University-Carbondale and a Bachelor of Science degree from Saint Louis Christian College.

Reviewers

The author and publisher wish to thank the following industry and teaching professionals for their valuable input into the development of *GD&T: Application and Interpretation*.

Sharon Bagby

Senior Instructor
Texas State Technical College
Abilene, TX

Debra Boren

Professor
Grayson College
Denison, TX

Anthony Cimabue

GD&T Specialist
Santa Fe, NM

Mark Durkee

Program Director
Madison Area Technical College
Madison, WI

Jonathan L. Gaddis

Drafting Technology Instructor
Southeastern Community College
West Burlington, IA

David L. King

Lecturer, Trainer, and Consultant
Technical Training Service
Clinton, CT

Tom Spendlove

Assistant Professor of Engineering
Baker College of Flint
Flint, MI

Acknowledgments

The author would like to express his appreciation to the following company for its cooperation in the preparation of this textbook. Parts shown in

photographs were provided by Technical Documentation Consultants of Arizona except where noted otherwise.

Technical Documentation Consultants of Arizona
2775 N. Shannon Road
Tucson, AZ 85745-1014

Photographs and illustrations are by Bruce A. Wilson unless noted otherwise.

Student Materials

- **Study Guide.** The Study Guide provides valuable student practice with questions and activities. Each chapter corresponds to the text and reinforces key concepts.
- **Online Textbook.** An online version of the printed textbook is available at www.g-wonlinetextbooks.com.

Instructor Materials

- **ExamView® Assessment Suite.** Quickly and easily prepare, print, and administer tests with the ExamView® Assessment Suite. With hundreds of questions in the test bank corresponding to each chapter, you can choose which questions to include in each test, create multiple versions of a single test, and automatically generate answer keys. Existing questions may be modified and new questions may be added.
- **Instructor's Presentations for PowerPoint®.** These presentations are designed to allow for customization to meet daily teaching needs.
- **Instructor's Resource CD.** One resource provides instructors with time-saving preparation tools such as answer keys, recommended activities, and correlation charts.
- **Online Instructor Resources.** Anywhere, anytime access to all instructional materials is convenient with this resource. Included in the Online Instructor Resources are the Instructor's Resource CD, Instructor's Presentations for PowerPoint®, and ExamView® Assessment Suite.
- **G-W Online.** G-W Online provides the opportunity to extend learning beyond the classroom. It enhances your course with course management and assessment tools that accurately monitor and track student learning. The ultimate in convenient and quick grading, G-W Online allows you to spend more time teaching and less on administration.

Contents

Chapter 1

Introduction to Dimensioning and Tolerancing 9

The Development of Dimensional Control	10
Design through Teamwork	14
Professions and Trades Affected by Dimensions	14
Required Dimensioning Skills	15
Orthographic Projection	18
Future Trends	20

Chapter 2

Dimensioning and Tolerancing Symbolology 23

General Symbols and Abbreviations	24
Geometric Tolerancing Symbols	29
Feature Control Frames	32
Datum Identification	34
Basic Dimensions	36
Past Practices	37

Chapter 3

General Dimensioning Requirements 43

Dimensioning Methods	44
Location and Size Dimensions	44
Line Use	45
Character Size and Style	47
Unidirectional Dimensions	48
Aligned Dimensions	48
Scale	49
Measurement System	50
Dimension Systems	53
Dimension Placement in Multiview Drawings	59
General Arrangement of Dimensions in Multiview Drawings	61
Dimensions Applied to Special Views	66
Dimensioning with Notes	68
Categories of Fit between Parts	70
Geometric Tolerances	72

Chapter 4

Dimension Application and Limits of Size 81

Dimension Application	82
Limits of Size	103
Calculation of Size Tolerances	105
Rules in ASME Y14.5	110
Dimensioning for Industry	113

Surface Texture Dimensions	117
Past Practices	120
Chapter 5	
Form Tolerances	127
Form Tolerance Categories.....	128
Feature Control Frames	129
Form Requirements from Size Limits.....	131
Straightness	133
Flatness	147
Circularity.....	152
Cylindricity	153
Chapter 6	
Datums and Datum Feature References	159
Datum Reference Frame.....	161
Datum Identification.....	163
Establishing Datums and Datum Reference Frames from Datum Features.....	168
Surfaces as Datum Features.....	170
Datum Features of Size.....	177
Datum Feature References in a Feature Control Frame.....	178
Special Applications	192
Advanced Datum Concepts.....	199
Chapter 7	
Orientation Tolerances	211
Orientation Tolerance Categories	211
Symbols and Application.....	211
Parallelism	216
Perpendicularity	220
Angularity	231
Tangent Plane.....	234
Combined Form and Orientation Tolerances	235
Chapter 8	
Position Tolerancing Fundamentals	241
Position	242
Effect of Material Condition Modifiers	246
Tolerance Calculation	250
Verifying Position Tolerances	255
Advantages of Position Tolerances.....	258
Special Applications	262
Chapter 9	
Position Tolerancing— Expanded Principles, Symmetry, and Concentricity	271
Composite Position Tolerances	272
Functional Gaging of Position Tolerances.....	280
Position Tolerance on Multiple and Simultaneous Patterns.....	287

Symmetry	295
Concentricity	298
Past Practices	299

Chapter 10

Runout 305

Circular Runout.....	305
Total Runout.....	313

Chapter 11

Profile 321

Profile Specification.....	321
Achievable Levels of Control.....	330
Past Practices	340

Chapter 12

Practical Applications and Calculation Methods 345

Floating Fastener Condition	345
Fixed Fastener Condition.....	351
Multiple Parts in Assembly	357
Zero Position Tolerance at MMC	362
Position Tolerance at LMC.....	364
Paper Gaging Techniques.....	364
Selection of the Applicable Tolerance and Modifiers.....	368

Appendices

A1—Coordinate-to-Diameter Conversion Table	377
A2—Coordinate-to-Diameter Conversion Chart	378
A3—Drill Size to Decimal Inch Diameter Table	379
B1—Current Symbols	381
B2—Symbols History	382

Glossary 384

Index 395

Chapter 1

Introduction to Dimensioning and Tolerancing

Objectives

Information in this chapter will enable you to:

- ▼ Explain the importance of accurately specifying dimensions and tolerances.
- ▼ Recall the history and development of dimensioning and tolerancing methods.
- ▼ Explain how teamwork can result in better definition of the dimensions and tolerances shown on a drawing or in a computer-aided design (CAD) file.
- ▼ Recall the job titles of those who should be on the design process team.
- ▼ Recall the dimensioning and tolerancing skills needed for success in design- or production-related occupations.
- ▼ Analyze some possible industrial changes and the impacts of these changes on dimensioning and tolerancing.
- ▼ Understand how views are created using orthographic projection.

Technical Terms

Anglo-Saxon Foot
British Imperial Yard
first angle projection
geometric dimensioning and tolerancing (GD&T)
Olympic Foot
orthographic projection
positional tolerancing
projection lines
Roman Foot
Royal Cubit
sixteen digit foot
third angle projection
tolerances

US Prototype Meter number 27
yard

Introduction

This highly competitive, industrial world requires full advantage be taken of all methods that help improve efficiency and reduce product cost. Application of dimensions and tolerances according to the current standards permits clearer definition of requirements than was ever possible before. Current standards define methods for increasing tolerance zones without reducing product quality or design function. Improving drawing clarity and increasing allowable tolerances are two means for improving competitive position.

A thorough knowledge of dimensioning and tolerancing methods helps to ensure clear application of part requirements. Tolerances may be maximized through careful dimension and tolerance calculations and application of the calculated values through proper utilization of standardized methods.

Improvements in tolerance specification benefit manufacturers of parts because they can read the drawing and determine exactly what is needed. This enables the manufacturer to match the appropriate process to the specified requirements. A clear definition of tolerance requirements is essential if parts are to be produced by more than one manufacturer. In many industries, including automotive and aerospace, parts are often made by supplier companies and shipped to a factory for assembly into a deliverable product. Requirements must be clearly defined if the parts are to fit together and properly function. Vague or incorrectly specified dimensional requirements can result in parts that are incorrectly manufactured, and ultimately unable to be fit into the assembly.

The number of small tolerances on a part should be reduced as much as possible while still meeting the needed quality, and all unnecessarily small tolerances should be eliminated. Calculating all tolerances is one step toward ensuring that tolerance values are maximized. Maximized tolerances permit the use of less expensive machine processes and less restrictive controls during the fabrication of parts.

The Development of Dimensional Control

There are two aspects to the historical development of dimensional control. One aspect is the establishment of standards that define how dimensions and tolerances are shown on a drawing. The other aspect is the establishment of a standard unit of measurement. The development of dimensioning and tolerancing standards has occurred only recently when compared to the time period over which units of measurement were developed.

Dimensioning and tolerancing standards became relatively well defined during the past 100 years. Most of the progress in standards development took place after the mid-1940s. The 1966 and 1973 American standards introduced far reaching new methods to clearly define tolerance requirements. Subsequent standards advanced capabilities, but the fundamental concepts have changed very little from the standard issued in 1973. Although fundamentals remained constant, the 1982, 1994, and 2009 standards introduced significant expansion of capabilities.

Measurement standards evolved over thousands of years, resulting in the two existing units of measurement—the inch and the millimeter. The measurement unit that is used has no effect on the manner in which the dimensions are applied; it only affects the numerical values that are shown in the dimensions and tolerances. The present dimensioning and tolerancing standard is applicable to the inch and the International System of Units (SI) metric dimensions.

Because either measurement system may be used, a note must be placed on each drawing to avoid any confusion as to which measurement system is in use. A drawing note such as the following may be used:

ALL DIMENSIONS ARE IN INCHES

or

ALL DIMENSIONS ARE IN MILLIMETERS

Measurement Standards

The existing measurement standards were developed over the past several thousand years. Units of measurement such as cubits, spans, palms, and digits were in use 5000 years ago. A *cubit* was the length of the Egyptian pharaoh's forearm. A *span* was the distance from the tip of the thumb to the tip of the small finger with the hand spread. A *palm* was the distance across the hand, and a *digit* was the distance across a finger. At the least, these standards were somewhat imprecise.

A permanent standard was made of black granite by the Egyptians. This was based on the cubit length (20.67") and was referred to as the *Royal Cubit*. It was subdivided into small increments to allow relatively short distances to be measured accurately. The Royal Cubit was a far more stable distance standard and was a reference against which measuring sticks were calibrated.

The Greeks established a unit of measurement called a *sixteen digit foot*. This distance standard (12.16") was approximately two-thirds the length of a Greek Cubit (18.93"). The official name for the sixteen digit foot is the *Olympic Foot*.

The Romans established a unit of measurement called the *Roman Foot*. It was 11.65" long. The *Anglo-Saxon Foot* was based on the Roman Foot and later became the standard for Great Britain.

The *yard* was established as a unit of measurement in the twelfth century when King Henry I of England decreed that the distance from his nose to the end of his thumb would be one yard. Three barley corns placed end-to-end were established as the standard for a one-inch increment in 1324 A.D. In 1855, a permanent standard for the yard was established in Great Britain and was called the *British Imperial Yard*.

In 1798, the standard meter was established. At the time, it was one ten-millionth of the distance from the North Pole to the equator when measured along a line passing through Paris, France. In 1889, the *US Prototype Meter number 27* became the standard against which all US measurements were to be referenced. Although the inch system is generally used in the United States, the inch length is referenced to the standard meter. The length of Prototype Meter number 27 is established relative to the length of a specific wavelength of light.

Evolution of the units of measurement from the length of the Egyptian pharaoh's arm to the wavelength of light has brought about a phenomenal increase in the accuracy of distance measurements. The evolution of a fixed and accurate distance standard allowed for improvement in

the equipment used to measure distance. See **Figure 1-1**. It would be impossible to produce machines that build and verify accurate distances without an accurate distance standard.

Dimensioning Standardization

It was not possible to verify the dimensions of a produced part before measurement standards were well established and an accurate means for gauging distances was created. All parts were fitted together and assembled by hand, because it was not possible to verify by measurement that separately produced parts would fit together. Drawings did not show dimensions with allowable variations (tolerances), because only crude means for verifying size existed. There was no reason to show limits of allowable variation on any dimensions because there was no means for accurately verifying size.

The development of accurate measurement equipment made it possible to verify relatively

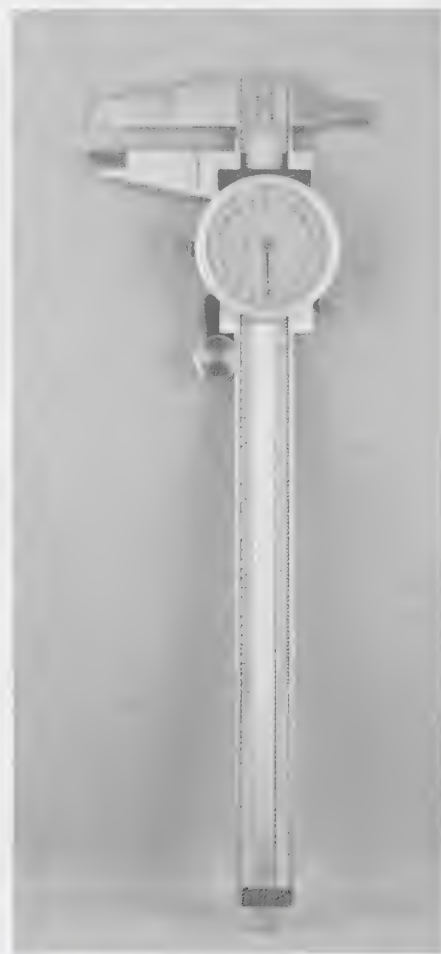
small variations in part size. See **Figure 1-2**. This capability makes it reasonable to specify the limits of acceptable variation for a dimension.

If limits of variation are correctly calculated, and all parts made within the specified limits, mass production is possible. Efficient mass production is only possible when any part made within the tolerances specified on a drawing can be assembled with its mating parts. This is not to say that each part manufactured to specifications is exactly alike, because variations will be present. The important point is that all variations are within a specified allowable range.

The repeatable accuracy that resulted from establishing a fixed measurement standard helped to make interchangeable parts possible. Eli Whitney is often credited with making the first interchangeable parts (1798). Repeatable accuracies that were achieved in the eighteenth century seem crude by today's capabilities, but they were adequate for making some interchangeable parts.



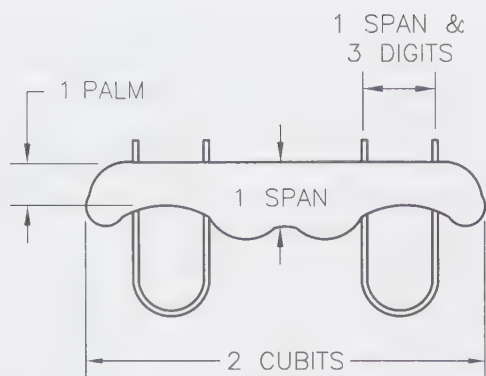
Scale and outside caliper



Dial caliper

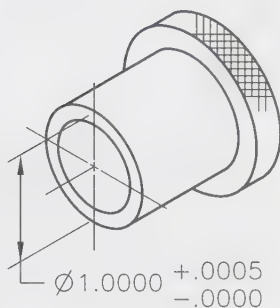
Goodheart-Willcox Publisher

Figure 1-1. A simple caliper and scale are shown on the left. As measurement standards evolved, so did the sophistication of measurement tools, such as the dial caliper shown on the right.



Wooden yoke

Built to crude distance standards to which tolerances would not be logical



Drill bushing

Built to accurate distance standards to which tolerances are logical

Goodheart-Willcox Publisher

Figure 1-2. Tolerances only make sense when there is an accurate unit of measurement.

Today, accuracies of .001 inch are produced for some part feature types, and some fabrication processes can achieve even greater accuracies. Present industrial capabilities make most any design feasible.

It is the responsibility of the person that applies tolerances to master the application guidelines. This allows for efficient utilization of today's industrial capabilities through properly specified dimensional requirements. However, there is a responsibility to utilize only the degree of dimensional control necessary to obtain the accuracy required for the part to function properly. Specifying accuracies smaller than needed increases both the cost and time required to produce a part.

The methods for completing design documents have advanced along with the capabilities of production and inspection machines. Drawing

instruments developed to the point where very accurate drawings could be produced, and computer-aided design (CAD) tools have now replaced drawing instruments in most companies. Computer-generated designs can contain dimensional data with accuracies far beyond what could be achieved with manual drafting practices.

Although computer-generated designs are extremely accurate, production machines cannot produce perfect parts from the CAD data. In the past twenty years, however, machines have improved and can achieve amazing accuracies. Regardless of the precision that is possible today, there is variation in every part fabricated. Because production machines introduce feature variation, it is essential that the amount of acceptable variation be defined. This is done through the application of tolerances. *Tolerances* are the acceptable dimensional variations that are permitted on a part.

Methods for showing dimensional requirements and permitted tolerances have advanced along with the measurement standards, drawing creation methods, and production machine capabilities. The dimensioning and tolerancing practices for showing size, form, orientation, run-out, location, and profile requirements are better defined today than ever before. The dimensioning and tolerancing practices are standardized to provide both a clear and consistent set of guidelines.

There has been a great deal of effort made to standardize dimensioning and tolerancing methods. The voluntary efforts of the people serving on national committees for standardization have resulted in the current standards. Their efforts are an important part of making mass production possible for the complex and accurate products in today's industry.

Dimensioning Standards

The main standard that defines dimensions and tolerances is ASME Y14.5. Other standards have some impact on what is shown in the dimensions, such as thread specifications, but ASME Y14.5 is the standard that shows what the dimensions and tolerances mean. ASME Y14.5 shows how the dimensions and tolerances are applied on 2D drawing views and to some extent on 3D models. ASME Y14.41 shows how they are applied in 3D views or CAD models (digital data files).

The methods for defining tolerances through what is commonly referred to as *geometric dimensioning and tolerancing (GD&T)* or *positional tolerancing* was begun in military documents

written in the 1940s and evolved into the MIL-STD-8 series of standards starting in the 1950s. The development of American industry standards began with USASI Y14.5-1966. The following list shows the evolution of the dimensioning and tolerancing documents.

MIL-STD-8A (1953)

MIL-STD-8B (1959)

MIL-STD-8C (1963)

USASI Y14.5-1966

ANSI Y14.5-1973

ANSI Y14.5M-1982

ASME Y14.5M-1994

ASME Y14.41-2003

ASME Y14.5-2009

A new issue of the dimensioning and tolerancing standard is released periodically. There is no fixed period of time between issues of the standard. New issues are released in an effort to keep the methods current with industrial needs. Longstanding practices are not changed unless there is a compelling reason to do so. The ASME Y14.5-2009 release made substantial advancements in the ability to define tolerances while maintaining the primary principles of previous standards. The advances are included in later chapters.

Seldom are existing practices reversed by a new issue. The new issues primarily add new methods and clarify those already in effect. The small number of changes and the time between issues of the standard are sufficient to reduce concern about the need to learn a new system every few years. Although changes are generally minimized, it is important to learn the changes made in a new issue of the standard.

It is sometimes necessary to know the content of the previous standards. This may be necessary when work is being completed on an existing drawing that was created in compliance with one of the older standards, or when creating a new drawing in fulfillment of an old contract that invokes one of the older standards.

Generally, it is not necessary to know the requirements of all previous standards because it is unlikely that many people are still working with the initial standards, such as MIL-STD-8. In the event that an old standard is a contractual requirement, a copy can be obtained from the contracting agency.

This textbook includes descriptions based on the current dimensioning and tolerancing standard and highlights areas of significant change from previous issues of the standard. The most recent issue of the standard should be used when there is an option regarding which issue (year

of standard) to use. If there is any doubt about whether a new issue of the standard has been released, the American Society of Mechanical Engineers (ASME) should be contacted.

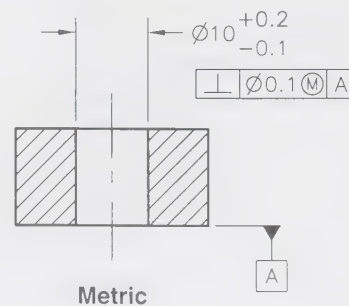
Standards are established to provide consistency in how things are done. However, a standard cannot show every possible situation that may arise; it can only show the principles and how they apply to various situations. It is up to the individual to apply the principles to specific situations as they are encountered.

Almost all figures in the current standard show metric dimensions. See [Figure 1-3](#). The only exceptions are figures that show how dimension limits are applied to inch values. *The standard does indicate that either inch or millimeter dimensions are permitted on drawings.*

A significant part of the industrial world still uses inch dimensions, especially in the United States. The illustrations in this book are shown mainly with inch values. Changing the inch values to millimeters would have no effect on how the dimensions and tolerances are applied; it would only change the numerical values and how they are displayed.

Principles covered by the current dimensioning and tolerancing standard may be applied to many situations that are not shown in the standard. If a complete understanding of the principles is gained, then the principles are adequate for meeting most design documentation needs.

Unique situations occur where the principles of the standard do not meet the needs for a specific dimension or tolerance application. In these cases, it is possible to use drawing notes to explain the special requirement. ASME Y14.5-2009 permits the use of notes to specify geometric tolerances, but that practice is not recommended as the general means of specifying tolerances. The use of notes to explain tolerance requirements should be avoided because one of the purposes of the current standard is to provide a means for clear definition of dimensional



Goodheart-Willcox Publisher

Figure 1-3. Illustrations in the current standard include metric dimensions.

requirements through symbology. A tolerance specification that correctly applies the symbology has a well-defined meaning. When notes are used, the clarity of the meaning may be damaged by a poorly worded note, or the note's meaning misunderstood by the person who reads it.

Design through Teamwork

Product cost can be minimized by involving people from various disciplines in the design process. Many designers are very knowledgeable about manufacturing and inspection processes; however, they may not know all the specifics about the operations. Teaming with manufacturing engineers, machinists, tool designers, tool fabricators, and inspectors is important in making sure the design can be produced at the lowest possible cost while still retaining the intended design function and ease of assembly.

Collaboration with the previously mentioned disciplines can help in many ways. Basic design concepts can often be improved through ideas from people who must operate the machines to produce the parts. These people know the capabilities of the machines. Designers usually know machine capabilities and processes, but it is difficult to be as well informed as the machine operators. Technology changes too fast for one person to remain fully knowledgeable about everything in all fields affecting a design. People who work full time in a discipline can provide accurate and valuable advice.

Designers must determine how to use the advice to create the optimum design that meets the functional requirements of the product. Part of the design optimization process is to make decisions about how to show the dimensional requirements and the tolerances. Once again, input from the previously mentioned disciplines can be very helpful.

There are significant advantages to having all disciplines review drawings and design documents as they are completed. Comments can be acted upon before the documentation is finished. This prevents or reduces the need for changes after the design is released and production is started. Another important advantage is the fact that everyone involved in the design process feels the design belongs partly to them. This helps develop a positive attitude among those who must produce the parts.

The most important advantage for involving people from various disciplines in the

design process is that part cost will be reduced. The design will be producible, machine programming will be made easier through better applied dimensions, tool design will be facilitated because of better specification of functional relationships between features, and inspection will be better directed because of a clearer understanding of part requirements. These are only a few of the benefits that can be realized by designing through teamwork.

Sometimes, it is tempting to complete a design through an individual effort. The temptation is probably caused by an apparent increase in efficiency during the design process. However, this apparent improvement in efficiency may be misleading, especially when considering the time spent on changes that will occur after production has started. These changes will be necessary because of errors and oversights that an individual working alone is bound to make. It is important to stress that working with a design team does not always reduce efficiency. A design team can work together and be efficient. Efficiency can be achieved through close cooperation and a willingness to achieve a common goal. A close-working team will be efficient when compared to one individual trying to determine all design parameters without the involvement of subject matter experts.

Professions and Trades Affected by Dimensions

Designers are obviously affected by dimensions and tolerances because they must apply these requirements on the design drawings or in the CAD model. A thorough knowledge of standard practices is required to make sure the information shown on the drawing conveys what is actually intended. Careless or uninformed application of tolerances can result in part requirements that are overly restrictive, forcing the part cost to increase. Incorrect application of tolerances can also result in inadequate control, resulting in parts that will not assemble or will not function properly.

A complete understanding of how to apply tolerances results in design requirements that are applied in a clear manner, with meanings that are supported by a national standard. This is important no matter where parts are produced. However, the importance of clarity increases when parts are made by a company other than the one that created the design.

Manufacturing engineers and *machine planners* are often the people who decide how parts are

to be produced. They must review drawings and make decisions based on the tolerances shown on the drawing. The dimensioning and tolerancing knowledge of the manufacturing engineer can have a significant impact on whether or not parts are made efficiently and correctly. Inadequate knowledge can result in an overly cautious approach; therefore, processes that are more accurate than needed might be used. This increases the product cost. Inadequate knowledge can also result in requirements that do not receive enough attention during fabrication.

Machinists and *machine programmers* must operate and create the computer programs for the equipment that produces the parts. These are the people who take the idea from a piece of paper, or CAD file, and produce a real part. If they understand the requirements, they can produce the parts with the greatest amount of efficiency. If they do not understand the requirements, there is little chance of obtaining functional parts at the lowest possible cost. Misunderstood requirements can result in either excessive or inadequate care in the completion of fabrication steps.

Inspectors verify that parts are produced in compliance with drawing requirements, including dimensions and tolerances. If inspectors have a good understanding of dimensioning and tolerancing principles, they will accept good parts (those within allowable tolerances) and reject bad ones.

Tool designers must design the jigs, fixtures, and other machining tools that are required to produce parts. Proper understanding of the dimensions and tolerances is required to correctly design the tools. Tolerance specifications often dictate how a part must be located in a machining tool. If the tool designer does not understand the tolerances, incorrect location methods might be designed into the machining tool. This could result in ruining all parts produced with the tool. Correct understanding of the requirements improves the chances that the tool design will be correct.

Fabrication of a good-quality product depends on a knowledgeable team. It is important that all people involved in the design and fabrication process work together to develop a full understanding of the standard practices. A knowledgeable team can produce a product of which they can be proud.

Knowledge of standard principles and careful application of that knowledge can also reduce the risks of product liability. Product liability should be a concern to all who work on a design or its production. Individuals as well as their companies are sometimes held responsible for errors.

Government and military design personnel need to have a thorough knowledge of dimensioning and tolerancing principles. Nearly all government design and production contracts require that drawings be completed in compliance with the current dimensioning and tolerancing standard.

The requirement for compliance with the dimensioning and tolerancing standard on government and military contracts also indicates the need for people working in the defense industry to learn the principles. Drawings and CAD models completed in compliance with the standard will be accepted with less need for revision than if there is incomplete compliance. Revisions that are necessary because of noncompliance with standards are usually made at the contractor's expense. If the expense of corrections must be paid by the contractor, that expense reduces the amount of profit made on the contract.

Proper application of the dimensioning and tolerancing principles will make part requirements more clear and result in tolerance zones that are determined by design function. Both of these factors reduce part production costs.

Required Dimensioning Skills

Dimensions and tolerances must be correctly applied to drawings and CAD models. This requires that standard application methods and symbology be understood. The ability to interpret the symbols applied to a design, as well as their meanings, is necessary because the parts must be produced according to what is specified on the drawing or CAD model. The ability to explain dimensioning and tolerancing requirements is needed when others who are less knowledgeable need assistance.

The ability to apply the tolerancing symbology on a drawing is only part of the needed tolerancing skills. It is also necessary to be able to calculate tolerances. Showing tolerances on a drawing is beneficial only if the tolerance values are correct. The only way to be sure tolerance values are correct is to determine the values through calculations. Other means of assigning tolerance values, such as similarity to previous designs or determination by judgment, introduces the risk of tolerances that may not result in the desired fit and function of the parts.

Application

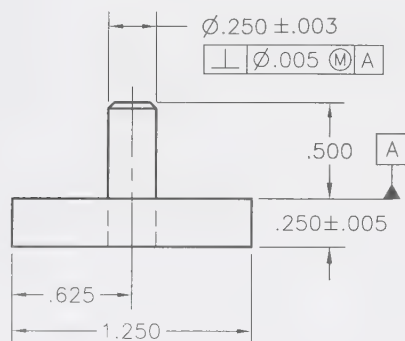
Decades ago, the methods of showing tolerances were limited to tolerances applied directly

to numerical values (such as size dimensions) and geometric controls that were explained in notes. It was not until the introduction of standard symbols that tolerancing methods progressed beyond the ambiguities of notations. The past practice of using notes to specify geometric requirements often resulted in uncertainty about the meaning of the note.

Symbols shown in the current dimensioning and tolerancing standard provide a means for defining requirements and remove the ambiguity that can result from using notes to define geometry. **Figure 1-4** shows a perpendicularity requirement for a dowel pin. This tolerance specification has a well-defined meaning (as will be shown in the chapter on orientation tolerances). The same control expressed through a note takes more room and is subject to the language ability of the writer.

Proper application of dimensions and tolerances is usually the responsibility of the designer or drafter. Therefore, he or she must have a good knowledge of dimensioning and tolerancing application principles. Various others may review the design documentation before it is approved for release and those reviewers are expected to ensure the documentation is correct. The reviewers must therefore have an understanding of the principles applied in the documentation for the review to be valid and reliable.

Persons applying dimensions and tolerances must use standard symbols in compliance with the methods defined in the applicable release of ASME Y14.5. The standard symbols are important where it is desired that others understand the information shown on the drawing. Nonstandard symbols should not be used, because nonstandard symbols have no defined meaning. Therefore, various opinions can be developed about the meaning of any nonstandard symbols. This can be somewhat alleviated by documenting a definition for any nonstandard symbol, and some large corporations do



Goodheart-Willcox Publisher

Figure 1-4. Symbology is used to define tolerances.

develop their own company standards that expand on the contents of the ASME standards.

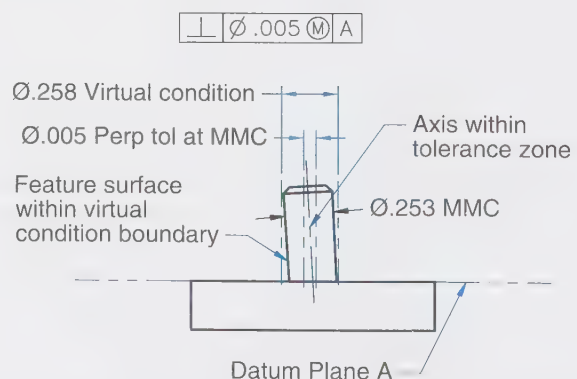
Interpretation

Parts are usually manufactured by someone other than the person that completed the drawing. In large companies, many people will be involved in making a part. Each of them completes some portion of the part requirements. This requires that each person understand the meaning of the drawing. See **Figure 1-5**. People need to develop enough knowledge of dimensioning and tolerancing principles to be able to efficiently interpret and understand a drawing.

Correct interpretation of dimensions and tolerances is required to produce a good-quality part at the minimum possible cost. Through correct understanding of the drawing requirements, it is possible to match the machine processes to the specified accuracies. The correct processes ensure that excessive errors will not be made, nor will excessive control be used.

Competent designers produce drawings or CAD models that accurately show the design requirements for their parts. Any person working with these drawings must interpret them. Because some designs are complex, it may take extensive knowledge of dimensioning and tolerancing principles to properly interpret the design requirements.

Every specification shown on a complex part may not be directly supported by a specific figure or paragraph in the standard. A person must be able to interpret dimensions and tolerances when they are applied to parts that have greater complexity than the figures shown in the standard. This means that the principles in the standard must be understood well enough to extend them to complex situations.



Goodheart-Willcox Publisher

Figure 1-5. Permissible geometry variations can be determined if tolerances are properly interpreted.

Some industry drawings contain dimensioning and tolerancing errors, and, therefore, are not in complete compliance with the standard. One reason for incomplete compliance is that not all designers have a thorough knowledge of dimensioning and tolerancing principles. Many have started to learn the subject, but are still working to master it.

When design documentation errors are identified, a decision must be made—work can be completed while ignoring the errors, or corrections can be requested. Sometimes, it is necessary to work with incomplete or incorrect design documentation. This is a risky practice, however, requiring that a person attempt to guess what requirements the designer meant to show on the drawing. If a portion of the drawing or CAD model is inadequate, then parts made to the assumed requirements may not function correctly. Working with an incorrect drawing or CAD model puts at risk the full cost of parts that may not be acceptable due to someone else's assumption of what the requirements were meant to state.

The amount of risk taken when working on the basis of an incorrect document is dependent on the type of errors. A judgment must be made as to the severity of errors and the potential cost of incorrectly interpreting what is intended. It is always better to ask for the requirements to be corrected than risk manufacturing a large quantity of bad parts.

When there is doubt about the meaning of a design requirement, it is a good idea to discuss the design documentation with the person who made it. If the verbal requirements and the documented requirements do not correspond, the documents should be corrected to reflect the actual design needs.

The ability to interpret dimensions and tolerances is only an ability to understand what a drawing means when it is completed in compliance with the standard. The ability to interpret a drawing is not an ability to guess what an incomplete or inaccurate drawing means.

Discussion of Principles

Understanding the dimensioning and tolerancing methods well enough to apply and interpret them is very important. It is also important to be able to discuss the principles with other knowledgeable people. Care should be taken to discuss the principles and methods that are defined by the standard, and not to create new methods that conflict with those already standardized. This requires knowledge of the principles and methods, and it

also requires knowledge of specific terminology that is applicable to dimensioning and tolerancing.

There is a wide range of dimensioning and tolerancing capabilities throughout industry. Many companies have utilized the principles covered in ASME Y14.5 for years. Others have just started to learn them, and a few have yet to begin. Even within companies that have started to use the methods, there is a wide range of capabilities. An absence of adequate knowledge can result in excessive fabrication or assembly costs and possible confusion regarding part requirements.

Not everyone will have the same opportunities to work with the dimensioning and tolerancing principles, and, therefore, will not learn them at the same rate. Each person will encounter different applications, may understand how to apply the principles to some applications, and may not fully understand others. Regardless of the reasons, various levels of capability do exist.

It is likely that anyone using the ASME Y14.5 principles will need to be able to explain what he or she has done. This is especially true when it comes to working with complex parts. It is also necessary to be able to understand the explanations of others as they describe new situations.

Calculations

Properly calculated tolerances take advantage of the maximum amount of part variation that can exist, yet still ensure proper function of the part. Many tolerance calculations are simple to complete; others require a significant amount of thought and may require the utilization of variation analysis software. An explanation of various tolerance calculation methods is given in the following chapters of this book.

Properly calculating tolerances is an important part of ensuring product quality and minimum cost. If tolerances are properly calculated, parts made within specified values will fit together correctly and function as intended. If tolerances are calculated, the values are known to be the maximum permissible amount of variation that will meet the functional and quality requirements. The benefit of maximizing tolerances is that large tolerances typically reduce part cost.

When design schedules are rushed, there may be a tendency to assign tolerance values without completing calculations. This is a risky situation. Parts made to *assumed* tolerances will not function correctly if the tolerances are larger than those resulting from calculations. If the assumed tolerances are smaller than necessary, the parts will cost more than necessary. Product liability and the

competitive industrial market should deter anyone from guessing at tolerance values. Records of calculations should be maintained to provide evidence of a good design.

Calculated tolerance values give the designer confidence the design will work. Anyone that has ever assigned tolerance values by “feel” or “guessing” knows the type of anxiety that is involved. A designer’s career is directly impacted by how well that person’s designs work. If tolerances are guessed, the designer must wait until parts are fabricated to determine if they will function properly. If the guesses are incorrect, then the chances of the design working correctly are significantly reduced. The wait and the uncertainty can be avoided by properly completing the tolerance calculations.

Designs do not always function as intended by the designer. When the first units are built, the designer must figure out what to do if the design fails to assemble or operate correctly. If tolerances are calculated, there is one less possible cause for the problem.

Orthographic Projection

Many schools and much of industry now use solid models to communicate both shape and dimensional requirements. Because of the widespread use of CAD to create solid models, not everyone entering industry will understand the two-dimensional (2D) views that appear on a drawing sheet or in a CAD file. This section provides a brief look at the common methods used to create 2D views in industry.

When 2D views are manually created, the views are typically arranged to make it easy to relate one view to another. The process used to manually create multiple 2D views is called *orthographic projection*. When 2D views are created from a solid model, the view creation process is mostly completed by the CAD software. However, when a drawing sheet is created from the solid model, the views should be positioned as though they were projected manually.

A brief introduction to orthographic views is provided here. Understanding how to “read” the views is important to correctly understand GD&T as it is presented in this book, and also to understand the drawings and models encountered in industry. It is beyond the scope of this book to completely describe the drawing creation process. Drafting textbooks provide a thorough explanation of view creation and rules that must be followed in creating the views. The following information should be sufficient to understand the

tolerancing practices illustrated in the figures of this book.

Angles of Projection

There are two projection systems that may be used to establish a known relationship between the views of a drawing. These projection systems are known as *first angle projection* and *third angle projection*. The United States typically uses third angle projection. Many other countries use first angle projection. In both systems, the views appear as though the features on the part are projected onto a flat surface, such as a drawing sheet. Each view is drawn as though the part is rotated 90°, or the viewing line of sight is rotated 90°.

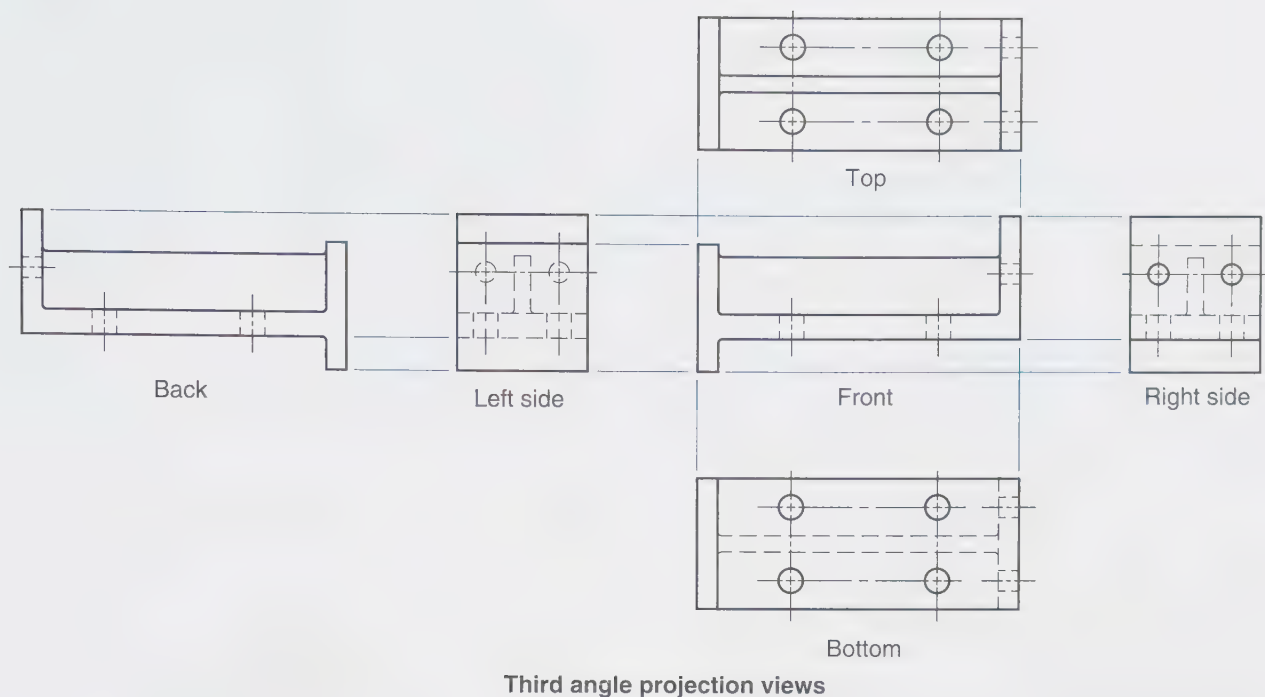
Relationship between Views

The view that shows the most detail about the part is usually selected as the front view. Once the front view is selected, the number of additional views needed to completely describe the part is determined. For some simple parts, it is possible to use only one view, but usually two or more views are needed.

Figure 1-6 shows the standard locations for six orthographic views that are drawn using third angle projection. For third angle projection, the example of a house makes the process fairly simple to understand. If the front of a house is shown in a front view, then it makes sense for the top of the house to be positioned above the front view of the house. Similarly, the right side of the house would be drawn to the right of the front view. Third angle projection works the same way for small parts. If you are looking at what seems to be the best front view of the part, rotating it to look at the top would create a top view that goes above the front view.

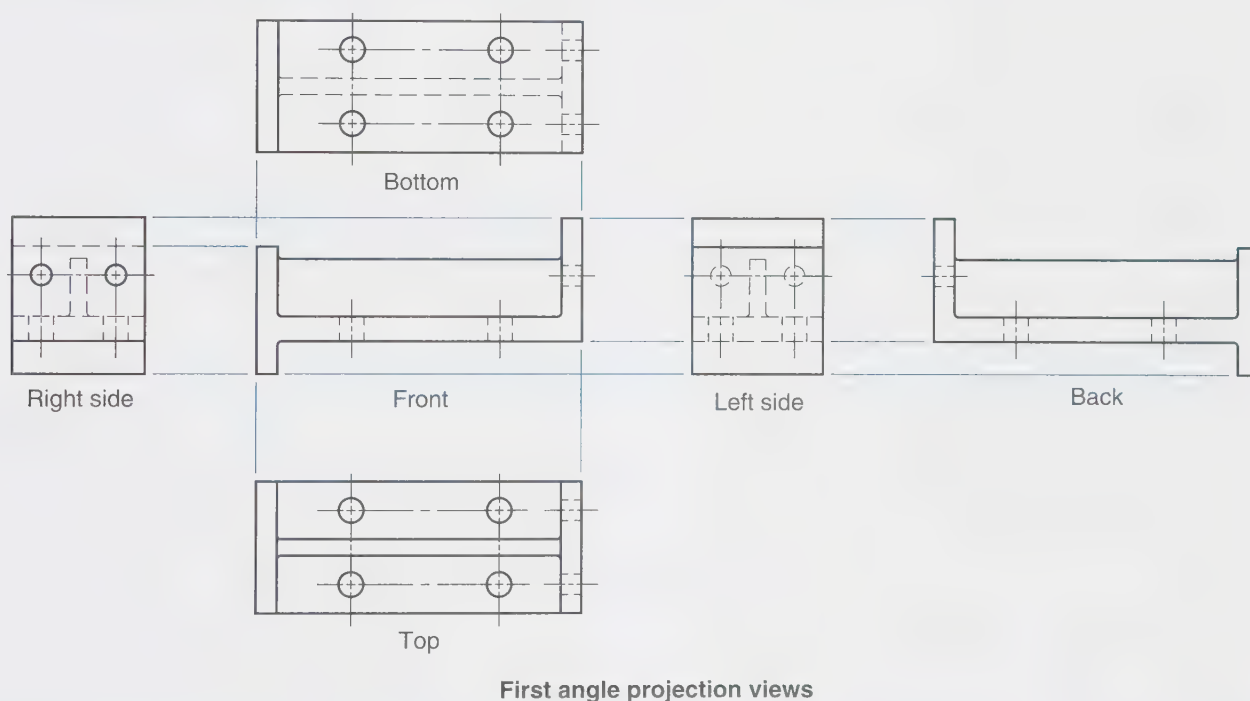
The part from **Figure 1-6** is shown again in **Figure 1-7**. However, the views are in positions based on first angle projection. What was the top view in the previous figure is now beneath the front view instead of above it.

It is important to determine if a drawing is created using first or third angle projection. The importance should not be overlooked. There are cases where the manufacturer incorrectly assumes the angle of projection. As a result, the parts are fabricated wrong. In one case, a structural part for a parking garage was fabricated using the wrong projection system, and the incorrectly made parts were put into the concrete columns before the error was noticed. Either the engineer made a mistake by not showing the applicable angle of projection symbol, or the fabricator read the drawing incorrectly.



Goodheart-Willcox Publisher

Figure 1-6. Orthographic views shown in third angle projection.



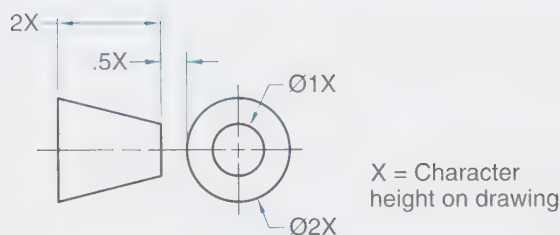
Goodheart-Willcox Publisher

Figure 1-7. Orthographic views shown in first angle projection.

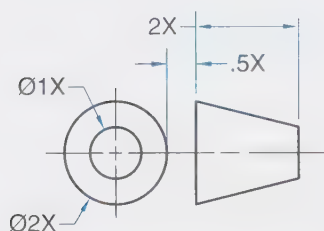
Regardless of who made the mistake, there was potential for loss of lives because of not understanding the angle of projection. With the global manufacturing that takes place, it is important

to avoid drawing interpretation errors that could potentially affect product quality and safety.

Figure 1-8 shows the symbols for third angle projection and first angle projection. Only one of



Third angle projection symbol



First angle projection symbol

Goodheart-Willcox Publisher

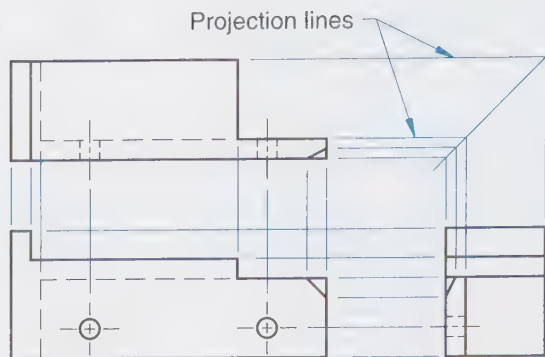
Figure 1-8. First and third angle projection symbols.

them should be used on a drawing. The projection symbol is typically shown in or near the title block of all drawings. Orthographic views in this textbook are drawn using the third angle projection system unless noted otherwise.

In orthographic drawings, each adjacent view appears as though the part is rotated 90°. Features in adjacent views can be related to one another by showing **projection lines** between the views. When manually created drawings are made, the projection lines are typically drawn very lightly so they do not appear in copies of the drawing. If all the projection lines were shown on a complex part, it would be very hard to tell the difference between the part geometry and the projection lines. When no projection lines are visible, it should still be possible to mentally connect the features in adjacent views. If the part is complex, it may be helpful to draw some of the projection lines. See **Figure 1-9**. The projection lines shown in this figure are not intended to be part of the final drawing but are included to show the relationship between features in the views.

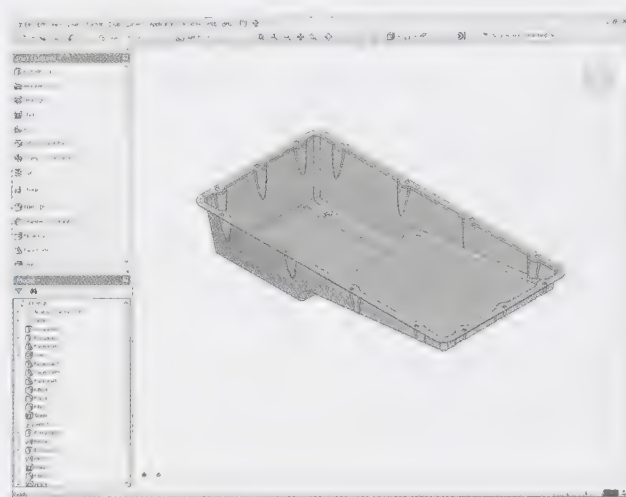
Future Trends

A significant change has taken place in industry over the past twenty years. The advancement and integration of software tools has resulted in replacement of manual drafting methods. Even small companies can now afford to use **computer-aided design (CAD)**. See **Figure 1-10**.



Goodheart-Willcox Publisher

Figure 1-9. Projection lines relate features in adjacent views.



Goodheart-Willcox Publisher

Figure 1-10. Manual drafting practices have generally been replaced by computer-aided design (CAD).

The minimum requirements for any CAD system are a computer and CAD software. A variety of computer models and software are available. The capabilities and prices of CAD systems vary. CAD systems range from inexpensive two-dimensional (2D) CAD software packages to highly advanced three-dimensional (3D) design systems.

A CAD system usually includes more than just the computer and software. It may also include output devices used to make **hardcopies** (paper copies) of the CAD design files. Software for operating **computer numerically controlled (CNC)** machines can also be utilized on the same computers as the CAD software. Variation analysis software may be integrated with the CAD software or operated as a stand-alone system. The variation analysis software is used to simulate the 3D variation effects, within an assembly, that result from detail part tolerances, tooling tolerances, assembly sequences, and assembly constraints.

Software for creating the computer files for running CNC machines is commonly known as *computer-aided manufacturing* (CAM) software. CAM files can be generated from CAD files. The designer creates CAD design files that define part requirements, and a manufacturing engineer or machine planner uses the CAD file to create a CNC machining program.

Some companies are now eliminating the need for hardcopies. The intent is to have the designer create a computer file that completely defines the part requirements. Someone else then copies the design file and creates the CNC machining program that runs the production machines. The CNC machining program is created without the use of a paper drawing.

A design file can also be used to create a computer file to operate an inspection machine. One type of inspection machine is called a *coordinate measuring machine* (CMM). One type of CMM is driven by a computer file called a CMM file. The CMM file is created from the design file and no paper drawing is needed. Creation of the design file, machining program, and inspection file can be completed without the generation of any paper. This type of operation is part of what is being referred to as the *paperless factory*.

The effect of the direct utilization of CAD files is that many companies now place their requirements directly in the CAD model and no drawing is created. Although some people still desire to see a hard copy in the form of a 2D drawing, the cost savings of directly applying requirements to the CAD model appears to be a driving factor. Manufacturing teams are learning to work directly from the CAD models and the trend is toward the elimination of 2D drawings.

People need to read dimensions only if they are going to manually input machining data for fabrication or if inspection is to be completed manually. Many companies have the ability to create CAD files, drive CNC machines with CAM generated files, and complete inspection of parts with CMM files, and they no longer need to have dimensioned drawings. The “as-designed” dimensions are defined by the CAD data, and only the tolerances are applied graphically. Some CAD systems are able to apply the tolerances graphically and also associate the requirements to the feature

parameters in a manner that other programs can access the information.

Regardless of whether the CNC and CMM machines get dimensions from a computer file or from manual input off a paper drawing, there must be some method for “telling” the machines how much variation is permitted. This means that tolerances must be applied to the CAD model in some manner so that downstream users of the design data know the part requirements.

CAD capabilities have already greatly reduced the number of paper drawings created. For many companies, these technologies have eliminated the need to show dimensions in the CAD file, but have not eliminated the need to indicate tolerance requirements and specifications.

The changes that the future holds will require a continual learning process for people to remain competent in their professions. Learning the information in this book will provide the knowledge necessary to work to the requirements defined by the current dimensioning and tolerancing standard and to prepare for the future.

Chapter Summary

- ✓ Development of an accurate distance standard was an important part of making possible the accurate production of parts.
- ✓ Compliance with dimensioning and tolerancing standards makes possible a common understanding of drawing requirements.
- ✓ A team consisting of people from various disciplines can produce a better design than an individual working alone.
- ✓ Many professions and trades are affected by dimensions and tolerances. The more knowledgeable people are about dimensioning and tolerancing, the more accurately and effectively they can complete their daily jobs.
- ✓ Required dimensioning and tolerancing skills include application, interpretation, and calculation. It is also an advantage if you are able to explain the meaning of various specifications on a drawing.
- ✓ CAD is the prevalent method of designing, dimensioning, and tolerancing industrial parts and assemblies.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. The current distance standard for the meter is based on the _____.
 - A. distance from the North Pole to the equator
 - B. wavelength of a certain color of light
 - C. distance between the earth's poles
 - D. distance around the equator
2. The ____ measurement unit must be used when working to the current dimensioning and tolerancing standard.
 - A. inch
 - B. millimeter
 - C. inch or millimeter
 - D. None of the above.
3. A(n) ____ must be able to understand dimensions and tolerances because that person is responsible for verifying that the produced parts meet the product requirements.
 - A. inspector
 - B. engineer
 - C. design manager
 - D. None of the above.
4. Producibility of a design can be improved the most by ____ working to optimize the design.
 - A. a single designer
 - B. a design team
 - C. two designers
 - D. None of the above.
5. Tolerance values should be _____.
 - A. assigned on the basis of what seems good
 - B. minimized to ensure proper function
 - C. calculated to ensure proper function
 - D. None of the above.

True/False

6. *True or False?* Measurements can be made more accurately when using the inch dimensioning system than when using the International System of Units.
7. *True or False?* If a symbol does not exist for a special need, it is a good idea to create one and use it on the drawing.
8. *True or False?* Standard tolerance application methods are required because inconsistent application methods would result in unclear requirements being shown.
9. *True or False?* Working from a drawing with incorrectly applied tolerances is likely to result in disagreements about whether or not the parts are properly made.
10. *True or False?* The top view is placed above the front view when third angle projection is used.

Short Answer

11. Why is it necessary to have an accurate distance standard in industry?
12. Cite one example of how a machinist can help a designer apply dimensions and tolerances in the best manner.
13. Show a general note that would be included on a drawing that primarily has millimeter dimensions on it.
14. Explain why an incorrectly applied tolerance can be a problem.
15. Why is it necessary for a tool designer to be able to interpret dimensions and tolerances?
16. Why is it important to know whether first or third angle projection is used to relate the views in an orthographic drawing?

Chapter 2

Dimensioning and Tolerancing Symbolology

Objectives

Information in this chapter will enable you to:

- ▼ Identify and draw general dimensioning symbols and show their general applications.
- ▼ Identify and draw tolerancing symbols and show their general applications.
- ▼ Complete a feature control frame using the correct order of segments in the frame.
- ▼ Identify basic dimensions and define means for indicating a basic dimension on a drawing or in a design model.

Technical Terms

all around symbol
all over symbol
angularity symbol
arc length symbol
basic dimension
circular runout symbol
circularity symbol
concentricity symbol
continuous feature (CF) symbol
controlled radius symbol
counterbore symbol
countersink symbol
cylindricity symbol
datum feature symbol
datum feature triangle
datum features
datum reference frame
datum target areas
datum target lines
datum target points
datum target symbol

depth symbol
diameter symbol
feature control frame
flatness symbol
free state symbol
independency symbol
irregular features of size
least material boundary (LMB)
location tolerances
maximum material boundary (MMB)
modifiers
movable datum target
origin symbol
parallelism symbol
perpendicularity symbol
position tolerance symbol
profile of a line symbol
profile of a surface symbol
projected tolerance zone symbol
radius symbol
regardless of material boundary (RMB)
regular features of size
slope symbol
spherical diameter symbol
spherical radius symbol
spotface symbol
square symbol
statistical tolerance symbol
straightness symbol
symmetry symbol
tangent plane symbol
taper symbol
total runout symbol
translation symbol
unequally disposed tolerance symbol
unilateral zone

Introduction

Symbology is used to express dimensioning and tolerancing requirements in engineering documentation, which includes engineering drawings and computer-aided design (CAD) models. The shapes of dimensioning and tolerancing symbols provide a logical connection between the symbol and the characteristic it represents. The easily identifiable shapes of the symbols permit a standard symbol set to be used internationally without the problems caused by differences in language.

It is important to create engineering drawings that have clearly defined dimensional requirements. The symbology defined by ASME Y14.5 makes a clear and consistent expression of requirements possible. The same level of clarity and consistency is difficult to achieve when notes are used instead of symbols. The difficulty with notations is caused by differences in language mastery levels and personal writing styles.

All tolerances are applied to part features, and part features fall into two major categories. They are either surfaces or features of size. As the meanings of tolerancing symbols are explained, some of them are limited to application on one of the two feature types. As an example, position tolerances may only be applied to features of size. Another example is the symbol for profile of a surface, and it may only be applied to surfaces. The meaning of a couple of tolerance symbols is altered depending on whether the symbol is applied to a surface or a feature of size.

There are two classes for features of size. *Regular features of size* are typically associated with a single size dimension such as thickness, width, length, or diameter. Regular features of size are made of common geometric shapes with a single size dimension such as a cylinder, circle, or slot. *Irregular features of size* may be one enclosed shape or a collection of features that in combination establish a size boundary. A hexagonal shape may be toleranced in such a manner that it is treated as an irregular feature of size. Throughout this text, the term feature of size is general and is applied to both regular and irregular features of size.










General Symbols and Abbreviations

Some dimension values shown on a drawing require clarification through the use of symbols. As an example, a dimension applied to specify

the diameter of a circular feature, such as a hole, includes a diameter symbol. The diameter symbol makes it clear that the specified value is a diameter, and not a radius or spherical diameter.

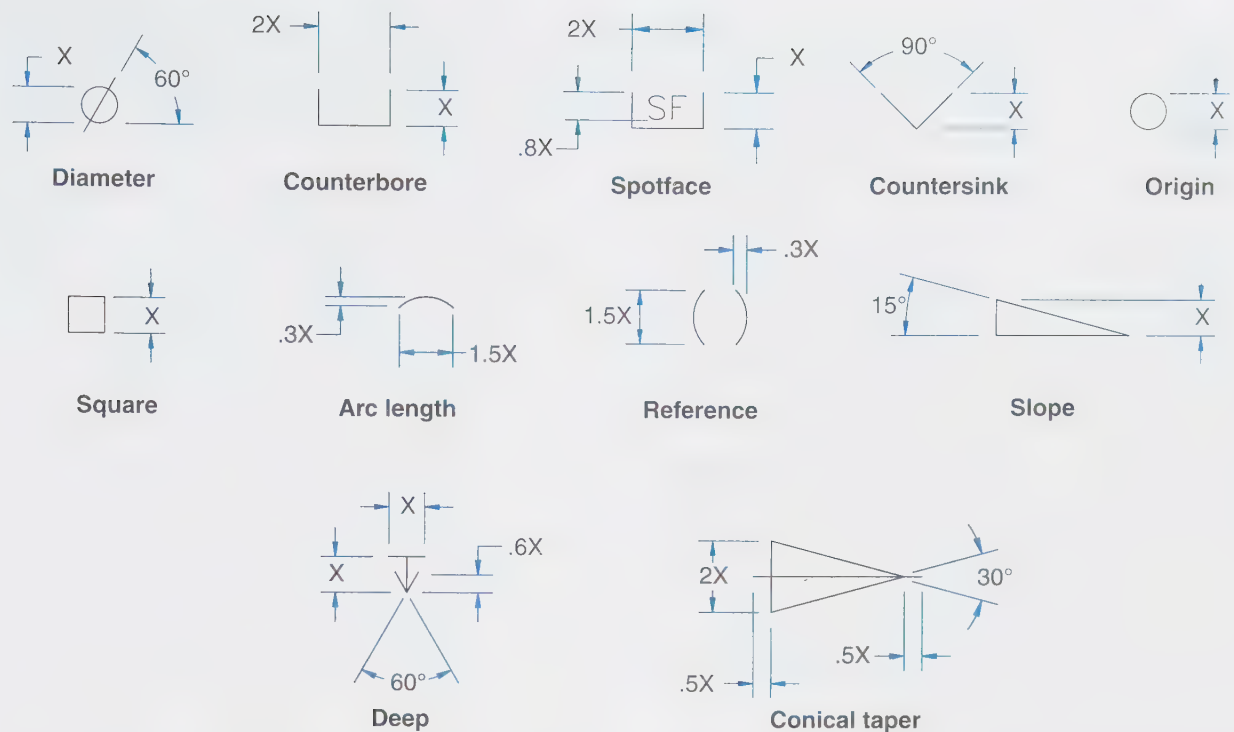
Standard dimensioning symbols, for characteristics such as a diameter, are used to provide consistency regardless of who creates a drawing. See [Figure 2-1](#). The symbols currently defined by ASME are shown. Where symbols are defined, abbreviations for equivalent terms are no longer included in the standard. The omission of abbreviations from the standard is a clear indication that symbols are preferred on the drawing or CAD model. The symbols may also be used in notes. However, abbreviations are commonly used in notes and this continues to be a fairly common practice because the symbols are not always available as a text font.

Symbol size on a drawing or CAD model is established relative to the character height of dimension numerals. See [Figure 2-2](#). The character height is to be substituted for the X in each of the symbol size formulas. The formulas shown in the figure are recommended symbol sizes and are not absolute requirements. Symbol size for use in notes is not standardized.

General Dimensioning Symbols		
Current practice	Abbreviation in notes	Parameter
∅	DIA	Diameter
S∅	SPHER DIA	Spherical diameter
R	R	Radius
CR	CR	Controlled radius
SR	SR	Spherical radius
	CBORE	Counterbore
	SFACE	Spotface
	CSK	Countersink
	DP	Deep
	—	Dimension origin
	SQ	Square
()	REF	Reference
X	PL	Places, times
	—	Arc length
	—	Slope
	—	Conical taper

Goodheart-Willcox Publisher

Figure 2-1. Symbology is used in place of abbreviations previously used for dimensioning.



Goodheart-Willcox Publisher

Figure 2-2. CAD software makes it easy to control symbol proportions. The proportions of symbols should be as shown in this figure.

The following example shows how the size of a symbol can be calculated.

Drawing character size = .125

Illustrated symbol dimension = 1.5X

Symbol size $1.5 \times .125 = .188$

Modern CAD systems include a library of dimensioning symbols and automatically size the symbols proportionally to the character size. To maintain uniformity, only one character size is used throughout a drawing or CAD model.

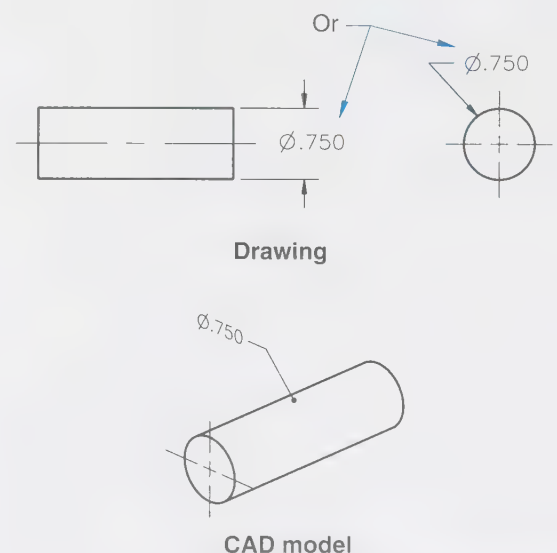
If you are using CAD software that does not contain the needed library of dimensioning symbols, a library of symbols should be created. A library of symbols permits the appropriate symbol to be pulled into a drawing without recreating the symbol each time it is needed. In the rare situation where drawings are created manually, templates may be used to draw symbols that are correctly proportioned.

Symbol Application

Symbol application requirements for drawings and CAD models are standardized in ASME Y14.5. Special application requirements for CAD models are defined in ASME Y14.41. Correct dimensioning practices defined by ASME Y14.5 and Y14.41 require placement of each symbol in a specific location relative to the associated numeric

value or relative to the feature to which the tolerance applies. Correct placement is introduced in this chapter and described in more detail throughout the rest of this book.

The *diameter symbol* is a circle with a line drawn through it. It always precedes the numeric value for the diameter. See [Figure 2-3](#). No space is shown between the symbol and the number. If



Goodheart-Willcox Publisher

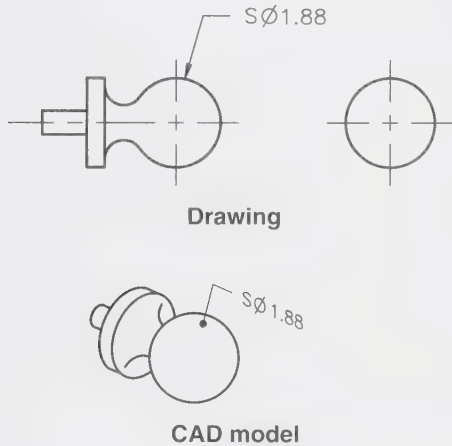
Figure 2-3. The diameter symbol immediately precedes the number.

a space were used, the symbol and number may appear detached. Past drawing practice used the abbreviation DIA following the dimension value.

Zeros should not have a line drawn through them to differentiate them from the letter O. A line through a zero can cause it to be mistaken for a diameter symbol.

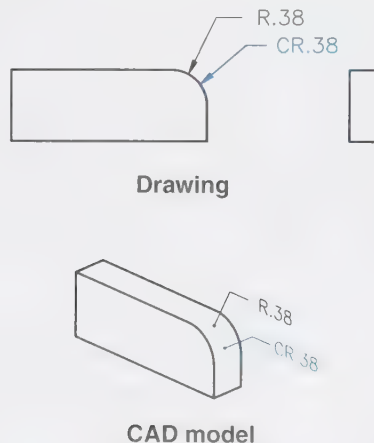
A *spherical diameter symbol* includes the letter S preceding the diameter symbol. The symbol is placed in front of the dimension value of the sphere. See [Figure 2-4](#). There is no space between the S and the diameter symbol and no space between the diameter symbol and the dimension value.

The abbreviation R is used as the *radius symbol*. The R precedes the numeric radius value. See [Figure 2-5](#). A space should not be shown between the symbol and number because the symbol would appear detached. Prior to the release of ANSI Y14.5M-1982, standard practice applied the letter R as a suffix to the radius dimension value.



Goodheart-Willcox Publisher

Figure 2-4. A spherical diameter is indicated with an S placed in front of the diameter symbol.



Goodheart-Willcox Publisher

Figure 2-5. The radius or controlled radius symbol precedes the number.

A *controlled radius symbol* is specified using the abbreviation CR as a prefix to the dimension value. It is applied in the same manner as the R symbol but includes a more exacting requirement on the feature surface.

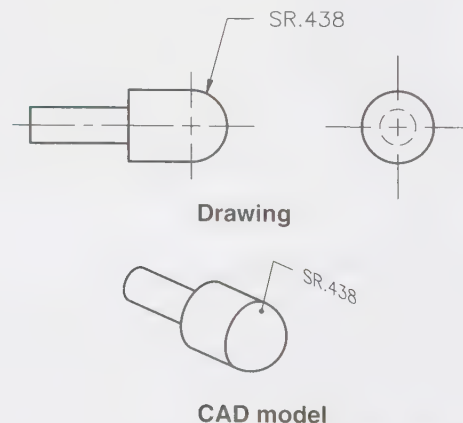
[Figure 2-5](#) shows two radius dimensions applied to the same surface. One has the R symbol and the other has the CR symbol. Both are applied in this figure to show the similarity in application. On a drawing or model, only one of the two dimensions would be applied based on the desired requirement for that surface.

A *spherical radius symbol* is indicated by the letters SR preceding the dimension value. No space is placed between the two letters. See [Figure 2-6](#). No space is shown between the letters and dimension value because the letters and numbers would appear detached.

Counterbores and spotfaces were, in the past, indicated by the same symbol, but they are now indicated by different symbols. The *counterbore symbol* is shaped like the bottom of a counterbore. See [Figure 2-7](#). The counterbore symbol precedes the diameter symbol and the numerical value representing the diameter of the counterbore. In the given figure, the counterbore diameter is followed by the depth symbol.

Depth may be specified in a dimension notation by placing the *depth symbol* in front of the depth value. The depth symbol is a downward-pointing arrow that extends from a horizontal line. See [Figure 2-7](#). To avoid confusion, the depth symbol should not be placed too close to the dimension value that precedes it.

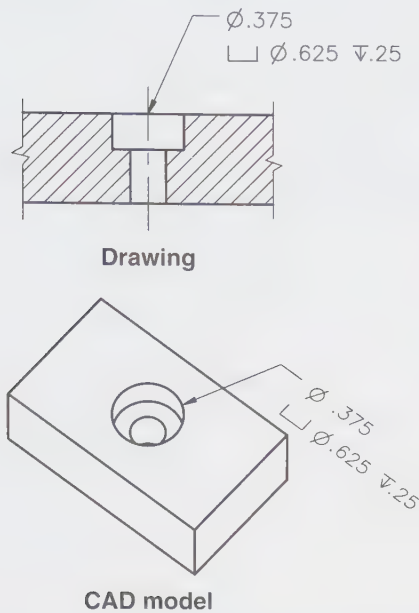
The *spotface symbol* is similar to the counterbore symbol except that it includes the letters SF inside the symbol, as shown in [Figure 2-8](#). The spotface symbol is followed by a diameter symbol



Goodheart-Willcox Publisher

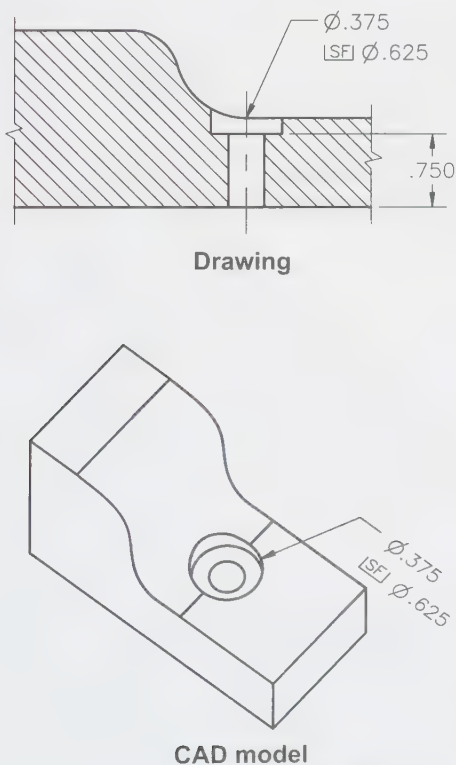
Figure 2-6. A spherical radius is indicated with an SR placed in front of the number.

and the diameter value. The diameter value is the size of the flat created on the surface and excludes any corner radius that may be included within the spotface. This is explained in greater depth in



Goodheart-Willcox Publisher

Figure 2-7. Counterbore, diameter, and depth symbols are used to specify the diameter and depth of the counterbore.



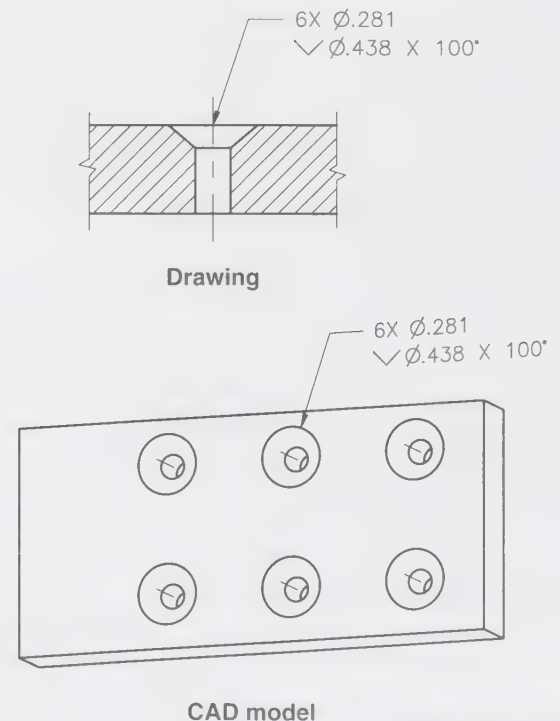
Goodheart-Willcox Publisher

Figure 2-8. The spotface symbol has the letters SF within the symbol that is used for a counterbore.

Chapter 4. The depth of the spotface may be specified in the notation or by a dimension, as shown in **Figure 2-8**. If the depth is not specified, the spotface depth is the minimum necessary to produce a flat surface across the entire diameter of the spotface. This is a typical process when a flat surface is needed on a rough casting. When the depth is not defined in a CAD model, the model may be queried to determine the depth.

The letter X may be used to indicate the number of instances, or places, a feature is required. See **Figure 2-9**. The countersunk hole shown in this figure is required in six places; the notation for the hole is preceded by 6X to indicate the required number of holes. There is no space between the 6 and the X. The meaning is changed if there is a space between the X and the preceding number.

The letter X may be used in a way that it means "by." It is used in this way when noting chamfer sizes and countersinks. See **Figure 2-9**. The countersink shown in this figure is noted to have a size .438" diameter by 100°. An X has the meaning of "by" when there is a space preceding and following (before and after) the letter. In the shown countersink notation, the letter X is used in one location to mean six times and in another location to mean .438" by 100°. Care must be taken to control the spaces around the letter X to ensure the proper meaning.



Goodheart-Willcox Publisher

Figure 2-9. Two applications and meanings for the symbol X and an application of the countersink symbol.

The *countersink symbol* is V-shaped; it has the appearance of a countersink. See [Figure 2-9](#). When specifying a countersink, the countersink symbol appears before the diameter symbol.

A dimension value enclosed in parentheses is a reference value. See [Figure 2-10](#). The proper usage of reference dimensions is explained in Chapter 3. If the reference dimension is a diameter or radius, it is optional whether the associated symbol may be inside or outside the parentheses. A past drawing practice used the abbreviation REF as a suffix to the reference dimension value.

The *arc length symbol* indicates that a given dimension defines an arc length. The symbol is an arc drawn above the dimension value. See [Figure 2-11](#). Past practice used a notation to indicate an arc length.

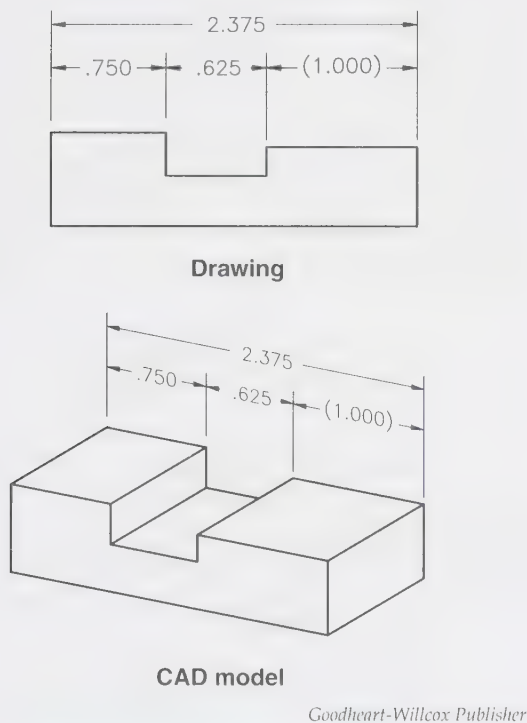


Figure 2-10. Parentheses around a number identify it as a reference dimension.

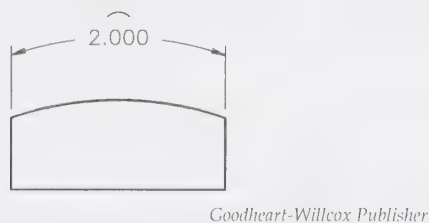


Figure 2-11. The arc length symbol is applied to indicate the length of an arc. Without the arc length symbol, the dimension could be mistakenly read as the width of the part.

A *square symbol* is used for dimensions that apply across the flats on a square shape. See [Figure 2-12](#). The square symbol precedes the dimension value. A space is not used between the square and the dimension value. Previous practice prior to 1982 used the abbreviation SQ as a suffix to the dimension.

An *origin symbol* is available for those applications where it is necessary to indicate the origin of a dimension. A small circle is placed at the terminating point of the dimension line that is to serve as the origin. See [Figure 2-13](#). The origin symbol replaces the arrowhead on the end of the dimension line that is at the origin.

The slope on a flat surface may be specified using the *slope symbol* and a slope value. See [Figure 2-14](#). The slope value is the amount of change in surface height per unit of distance along the base line from which height measurements are made. If the change in height is specified in inches, then the unit length is one inch. The slope symbol is placed before the slope value. A complete slope specification includes both the amount of slope and tolerance on the slope per inch along the base line. The slope specification is written as a ratio. The specified slope value in [Figure 2-14](#) indicates a rise of .20" plus or minus .01" per inch of distance (.20±.01:1).

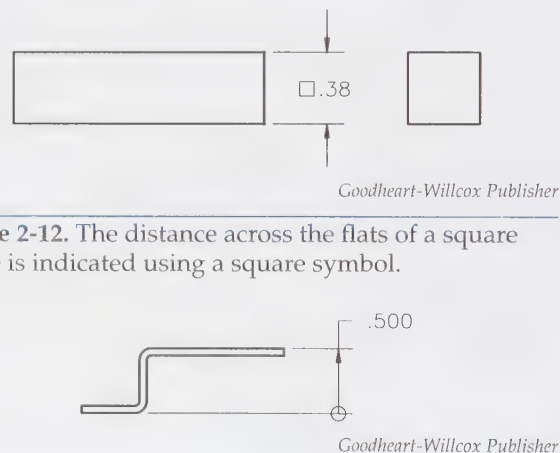


Figure 2-12. The distance across the flats of a square shape is indicated using a square symbol.

Figure 2-13. The origin symbol is used to indicate the surface from which measurements are to be made.

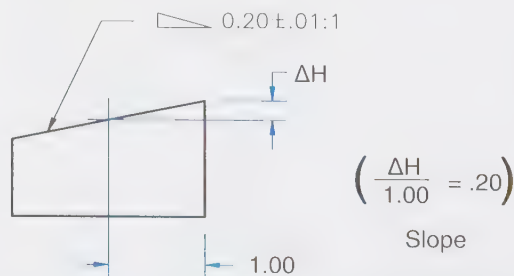


Figure 2-14. Slope symbol application.

Several methods exist for dimensioning tapered parts. When the amount of taper per unit of length is specified, a **taper symbol** is used. See **Figure 2-15**. The symbol appears as a cone on an axis. The taper symbol precedes the taper value and its tolerance. The specified taper value in **Figure 2-15** indicates a change in diameter of .18" plus or minus .02" per inch along the axis (.18±.02:1).

Geometric Tolerancing Symbols

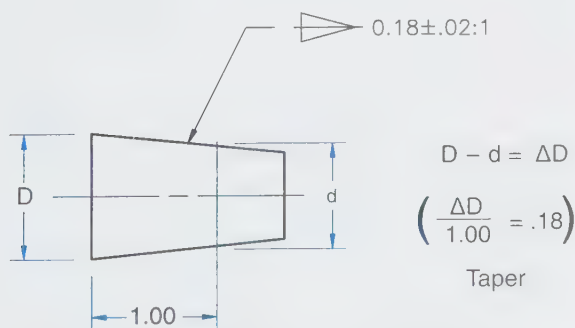
Tolerances are applied on drawings or in CAD models to define the allowable variations for the part features. Tolerances are required because it is not possible, efficient, or cost effective to produce every part to an exact size.

The utilization of symbology for the specification of tolerances has been evolving from a time before MIL-STD-8. Initially, control of form, orientation, and position was achieved through notes placed on the drawing. As the inconsistencies and vagueness in notations became an obvious communication obstacle, the utilization of symbols began to replace notations. Symbology is now utilized for a majority of specified tolerances. Only occasionally is a tolerance needed that requires notations to supplement the existing symbology.

Symbol Shape and Size

Tolerancing symbols, like general dimensioning symbols, have shapes that logically connect to the associated control. Correct utilization of standardized symbols provides a clear tolerance specification. Utilization of nonstandard symbols or misapplication of standard symbols can cause tolerances to be ambiguous.

Standardized symbols exist for specifying form, orientation, profile, runout, and location tolerances. There are additional symbols for clarifying



Goodheart-Willcox Publisher

Figure 2-15. Taper on a diameter may be dimensioned using the taper symbol.

and modifying tolerances. Each symbol has a recommended size that is related to the general character height used on the drawing or in the CAD model. See **Figure 2-16**. Illustrated symbol proportions include dimension values related to the variable X. The value of X represents the character height on the drawing or in the CAD model where the symbol is to be used.

Form tolerance symbols are used for specifying requirements that apply to individual features. In some special cases, multiple features may be noted as acting as one continuous feature. Form tolerances may be applied to those continuous features and treated as though only one feature exists, but this may create confusion and is not recommended. The application and interpretation of form tolerance specifications is explained in Chapter 5.

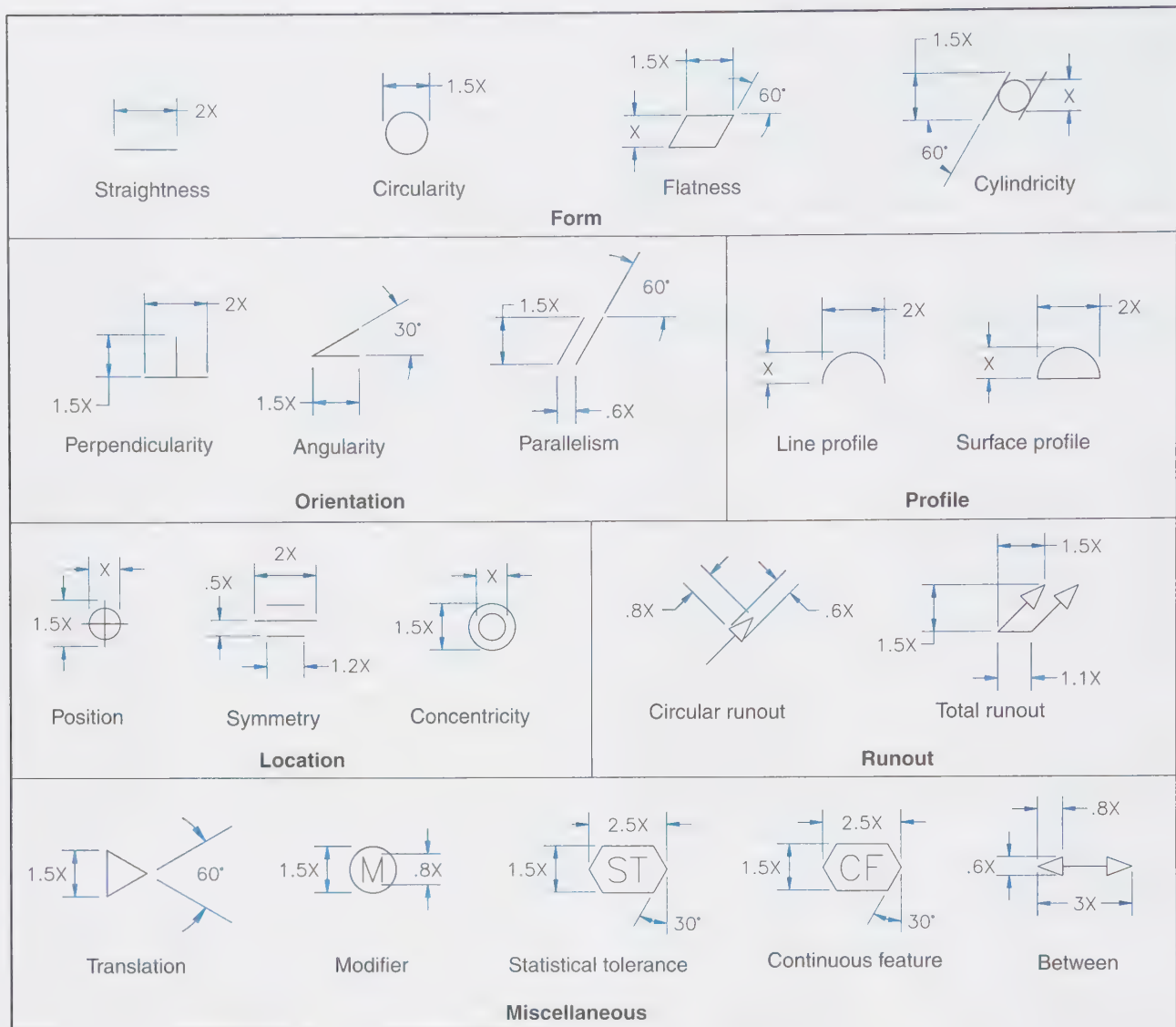
The form tolerance symbols are straightness, flatness, circularity, and cylindricity. The **straightness symbol** is a short, straight line. Because the symbol has the same shape as the desired control, it is easily identified. The **flatness symbol** is a parallelogram. This symbol can be thought of as an oblique view of a flat surface. A circle is used for a **circularity symbol**. As with the straightness symbol, this symbol resembles the indicated control. The **cylindricity symbol** is a circle with two tangent lines.

The symbols for specification of orientation tolerances are perpendicularity, parallelism, and angularity. See **Figure 2-16**. The **perpendicularity symbol** is indicated by two perpendicular lines. The **parallelism symbol** is indicated by two inclined parallel lines. The **angularity symbol** is two lines drawn to form a 30° angle.

There are two types of profile tolerances. See **Figure 2-16**. The **profile of a line symbol** (also called line profile symbol) is a semicircle. The **profile of a surface symbol** (also called surface profile symbol) is also indicated by a semicircle, but it has a horizontal line drawn across the bottom to distinguish it from the profile of a line symbol.

There are two types of runout tolerance symbols. See **Figure 2-16**. The **circular runout symbol** is a single arrow. The arrowhead may be filled solid, or it may be left unfilled. The **total runout symbol** is composed of two arrows connected by a horizontal line. The total runout symbol shown in the figure was first made a requirement in the 1982 issue of the dimensioning standard.

There are three types of **location tolerances** and each has its own symbol. They are position, symmetry, and concentricity.



Goodheart-Willcox Publisher

Figure 2-16. Tolerance symbols are shown grouped by tolerance categories.

The *position tolerance symbol* is composed of two lines crossing at the center of a circle. See Figure 2-16. It is symbolic of the center lines used on a drawing to indicate the desired location for a hole. Another type of location tolerance is concentricity. The *concentricity symbol* is indicated by two concentric circles.

A third type of location tolerance is symmetry. Symmetry may be indicated by either of two types of symbols, depending on the type of control desired. A position symbol may be used when specifying the symmetrical location of features of size at maximum material condition (MMC). A *symmetry symbol* that consists of three horizontal lines may be used for other symmetry tolerances. The symmetry symbol containing three lines was not included in the 1982 standard.

Modifiers are often applied to tolerance specifications. The proportions shown for modifiers in Figure 2-16 apply to material and boundary condition modifiers, the independency symbol, free state symbol, projected tolerance zone symbol, tangent plane symbol, and the unequally disposed tolerance zone symbol.

A double-ended arrow is used in place of the word *between*. It is used for tolerance specifications that extend between labeled points, edges, or other features.

A tolerance calculated by statistical methods may be identified with a special symbol. The *statistical tolerance symbol* is an elongated hexagon containing the letters ST.

The *continuous feature (CF) symbol* is an elongated hexagon containing the letters CF. This

symbol indicates that a dimension or tolerance applies across interruptions in a feature of size so that it acts as a single continuous feature. A continuous feature is sometimes referred to in industry as an *interrupted feature* or a *controlled feature*. Past practices sometimes used an extension line to indicate a continuous feature, but that practice is not supported by the current standard.

Other Symbols

Every geometric tolerance applied to a feature of size is applicable regardless of that feature's size unless a modifier is shown that requires the tolerance be applicable at a specific material condition. In a similar way, every datum feature reference is applicable regardless of material boundary unless the datum feature reference is modified to apply at a specific material boundary. The applicable material or boundary condition is specified with symbology shown in the tolerance specification (feature control frame), or it is implied on the basis of rules that are defined in later chapters.

There are two material condition modifier symbols defined by the standard, and they are used to indicate the material condition or the boundary condition that is applicable. The same symbol (letter M inside a circle) is used for both maximum material condition (MMC) and maximum material boundary (MMB). Similarly, one symbol (letter L inside a circle) is used for least material condition (LMC) and least material boundary (LMB). If neither of these modifiers is shown in a tolerance specification, the tolerance is assumed to apply regardless of feature size (RFS). There is no modifier symbol for RFS. If neither of the symbols is applied in a datum feature reference, the datum feature is assumed to apply regardless of material boundary (RMB).

Standards Advisory

MMB, LMB, RMB

The following was introduced in the ASME Y14.5-2009 standard. When the MMC and LMC symbols are used on datum feature references, their meaning is expanded to include *maximum material boundary (MMB)* and *least material boundary (LMB)*. MMB and LMB may be applied to datum feature references that are surfaces. Datum feature references without a material boundary modifier symbol are assumed to apply *regardless of material boundary (RMB)*.

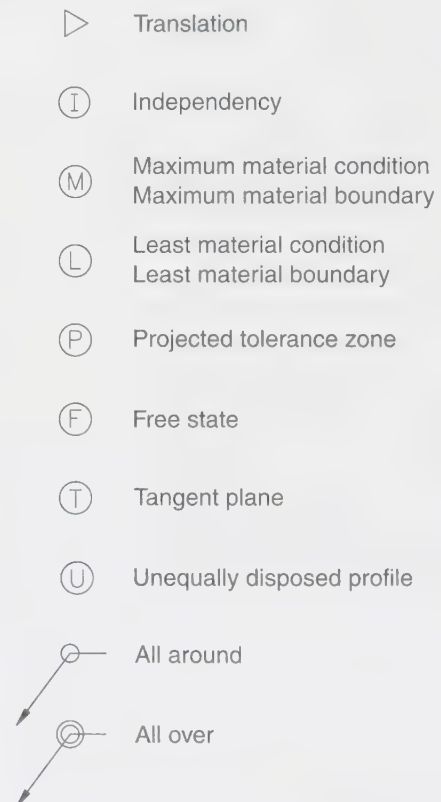
Material condition modifier symbols are used in tolerance specifications in CAD models and on drawings. They are not typically used in drawing notes or other written text because of font limitations. Where it is necessary to reference a material condition modifier in written text, the term may be spelled out or the appropriate abbreviation is used.

Some tolerance applications require the specified tolerance zone to project outside the object. When this is necessary, a *projected tolerance zone symbol* is used. See Figure 2-17. The projected tolerance zone symbol is the letter P within a circle.

Where tolerances applied on flexible parts are intended to apply in the free state, they are identified by the application of a *free state symbol*. The free state symbol is the letter F inside a circle.

Tolerances that apply to a tangent plane instead of to the surface are identified by the *tangent plane symbol*, a letter T inside a circle.

The letter U inside a circle is the *unequally disposed tolerance symbol*. This symbol is shown in a tolerance specification to indicate that a profile tolerance is unequally disposed. The unequally disposed zone may be a *unilateral zone* (entirely to one side of the basic profile) or a zone that is



Goodheart-Willcox Publisher

Figure 2-17. Modifier symbols are primarily used within feature control frames.

unequally offset relative to the basic profile. This is further explained in the profile tolerances chapter.

The letter I inside a circle is the *independency symbol*. Rule #1 of the standard requires that, by default, a feature of size must have perfect form at its maximum material condition. When it is desired to take exception to that rule, the independency symbol may be applied to the dimension value.

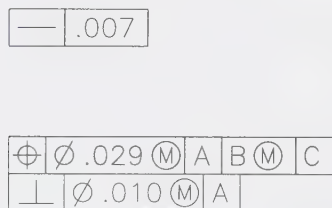
An equilateral triangle pointing to the right is the *translation symbol*. This symbol is used with a datum feature reference to indicate that the datum feature simulator may translate (move). Application is explained in later chapters.

Profile tolerances may be applied to extend all around the profile of the view to which the tolerance is applied. Multiple tolerance specifications may be avoided on a view by using an *all around symbol*. See Figure 2-17. The all around symbol is a circle placed at the corner in the tolerance application leader. It is similar to the all around symbol used in welding specifications. The *all over symbol* is two concentric circles located at the corner in the leader. This indicates the profile tolerance applies all over all the surfaces of the part.

Feature Control Frames

Geometric tolerance requirements are specified in a *feature control frame* or in notes. A feature control frame is a rectangle made up of compartments containing symbology and numerical values to define a tolerance requirement. Feature control frames may contain no more than a tolerance symbol and a tolerance value, or they may contain multiple lines of requirements that include tolerance symbols, tolerance values, modifiers, and datum feature references. See Figure 2-18. The amount of information contained in a feature control frame depends on the desired level of control to be established.

The following segments of this chapter describe the required composition for a feature



Goodheart-Willcox Publisher

Figure 2-18. A feature control frame may be composed of one or more lines. Each line is called a *segment* in ASME Y14.5.

control frame. Determining the appropriate information to include in a feature control frame is a primary subject of this book, and the methods for completing each type of specification are described in several chapters.

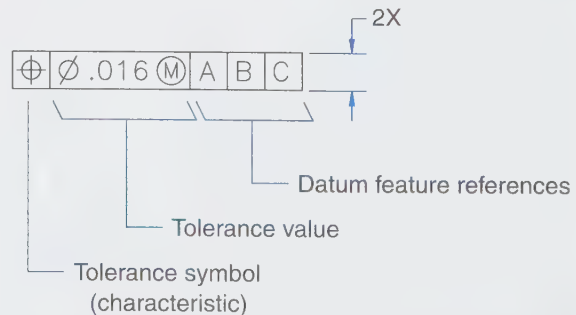
Composition of the Feature Control Frame

Every feature control frame follows the same order of specification, reading from left to right. See Figure 2-19. The tolerance symbol (characteristic) is always shown first at the left end of the feature control frame. It is followed by the tolerance value. Datum feature references, if required, follow the tolerance value.

Words are generally not written inside a feature control frame. Some special applications do exist, including the use of MAX, NONUNIFORM, and BASIC, but symbols, numbers, and datum feature reference letters are generally adequate for specification of standardized tolerance controls.

The tolerance value in a feature control frame may be no more than a number indicating the size of the tolerance zone. It is also possible for the tolerance value to include a diameter symbol, a spherical diameter symbol, a material condition modifier, a statistical tolerance symbol, and a tangent plane symbol. See Figure 2-19. When used, the diameter symbol precedes the tolerance value. It is required if the tolerance zone is round or cylindrical. A spherical diameter symbol precedes the tolerance value if the tolerance zone shape is spherical. When a material condition modifier is shown, it follows the tolerance value. The tangent plane modifier follows the tolerance value when it is used. (See Chapter 7.) The statistical tolerance symbol follows the tolerance value and any material condition modifier that is included.

Datum feature references are required in all feature control frames where the tolerance zone has a required relationship to one or more datums. The number of referenced datums depends on the



Goodheart-Willcox Publisher

Figure 2-19. Feature control frames are always read from left to right.

type of tolerance being specified and the control that is desired.

Datum feature references are always written left to right with the primary datum shown first. See [Figure 2-20](#). It is followed by the secondary and tertiary datum feature references. Material boundary modifiers may be applied to the datum feature references.

Placement of the Feature Control Frame

A properly composed feature control frame must be shown in the CAD model or placed on the drawing in the correct manner to ensure the appropriate requirement is expressed. Correct application will clearly indicate the feature or features that are toleranced. Some tolerance types may only be applied to surfaces and others only to features of size, so correct application is essential to avoid specification of meaningless or incorrect tolerances. Two tolerances, straightness and flatness, may be applied to surfaces or features of size. These two tolerances when applied to surfaces have a completely different meaning than associating the same tolerances with a feature of size.

Application to Surfaces

Placing the feature control frame on an extension line from a surface in an orthographic view indicates the specification of a requirement that is applicable to the surface. Feature control frames may be placed on either side of extension lines. The side of the extension line on which it is located makes no difference. See [Figure 2-21](#). Feature con-

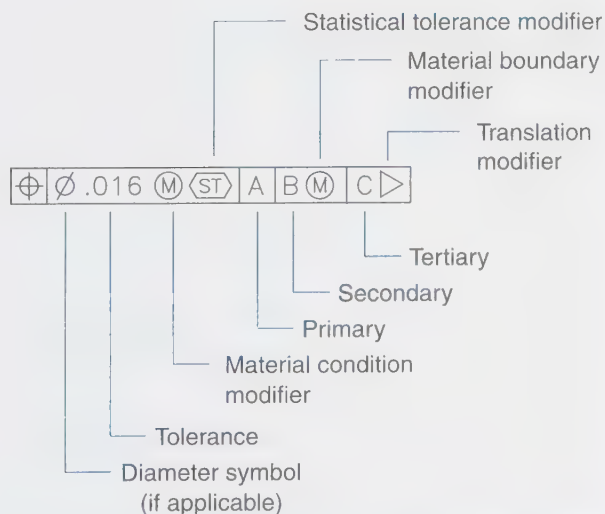
trol frames are generally not shown on extension lines in CAD models.

A leader may be used to connect a tolerance specification to a surface in orthographic views and in CAD models. The use of a leader has exactly the same effect as attaching the specification to an extension line. The leader may be extended from either end of the feature control frame.

The leader terminates on the surface in CAD models and ends with a dot on the surface. If a leader is used in an orthographic view, it may terminate with an arrowhead on a line representing the profile of the surface. In situations where no edge view of the surface exists in an orthographic drawing, the leader may terminate on the surface using a dot as shown in the CAD example of [Figure 2-21](#).

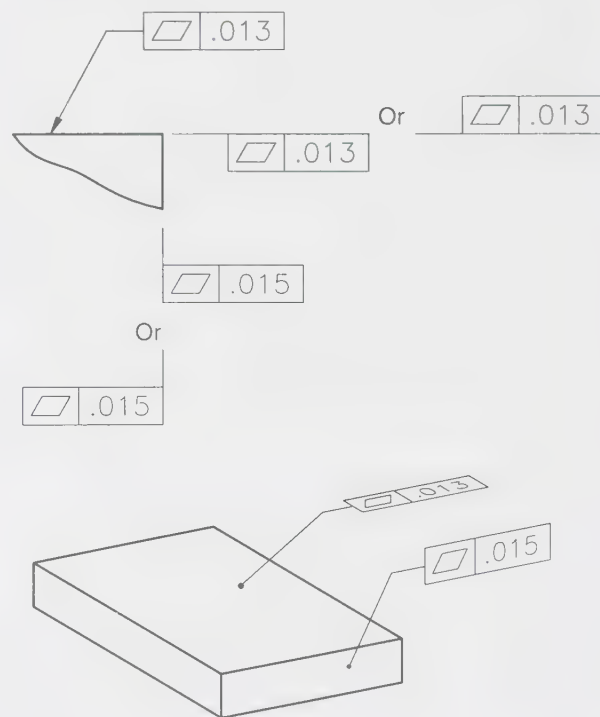
Application to Features of Size

Tolerances may be applied on a feature of size by placing the feature control frame adjacent to a dimension or attaching it to the dimension line. See [Figure 2-22](#). When a feature control frame is placed adjacent to a dimension value, care must be taken that confusion does not exist regarding other dimensions in the vicinity. The feature control frame is to be clearly associated with only one dimension.



Goodheart-Willcox Publisher

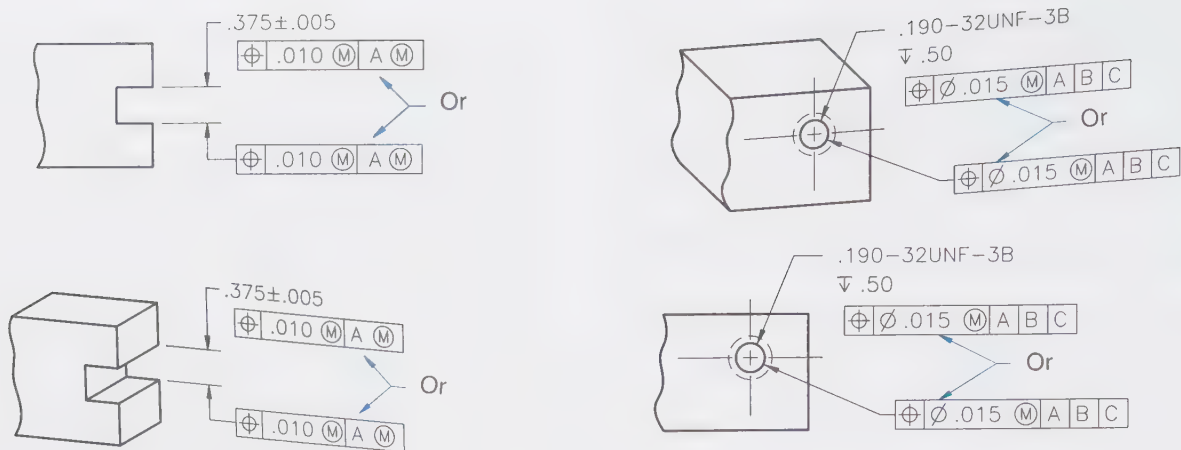
Figure 2-20. The desired tolerance specification and the type of feature being controlled determine when a diameter symbol or material condition symbol is required.



Application to surfaces

Goodheart-Willcox Publisher

Figure 2-21. The end of the feature control frame to which a leader or extension line attaches does not affect the interpretation of the requirements.



Application to features of size

Goodheart-Willcox Publisher

Figure 2-22. Placement of a feature control frame on the dimension line or near the dimension value indicates a control on the feature of size.

Application to Threads

A geometric tolerance applied to a thread is either placed adjacent to the thread specification or is attached to the threaded feature by a leader. See [Figure 2-23](#). Any tolerance in a feature control frame that is applicable to a thread is by default applicable to the pitch diameter and the resulting pitch cylinder axis.

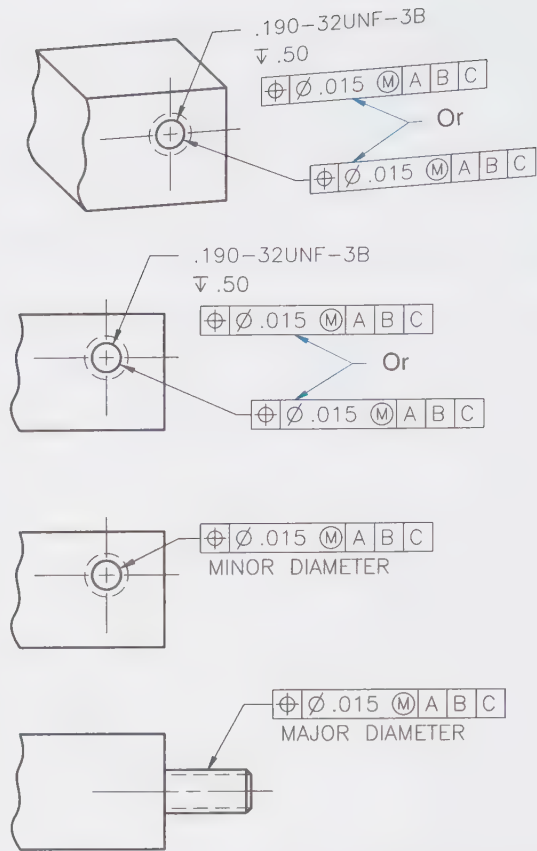
Tolerances applied to threads typically should apply to the pitch diameter. Sometimes it is desirable to specify a geometric tolerance that controls the major or minor diameter of a thread. This can be done by noting the applicable diameter beneath the feature control frame. See [Figure 2-23](#). The first two examples are applicable to the pitch diameter. This is the way that most threaded features are tolerated. The two exceptions shown in the lower portion of the figure show the tolerance noted as applicable to the minor diameter on an internal thread, and the major diameter on an external thread.

Application to Gears

Geometric tolerances applied to gears are not generally applied to the gear teeth. Past practices of applying runout tolerances to gear teeth have been replaced by other methods that are defined by gear specific standards. Any desired tolerances on gear features other than the teeth may be accomplished through application of appropriate feature control frames.

Datum Identification

Datum features are the surfaces and features of size used to establish theoretically perfect datum



Goodheart-Willcox Publisher

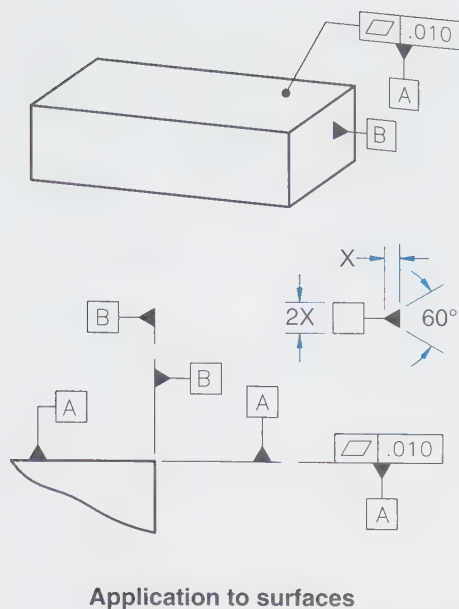
Figure 2-23. A notation must be placed under the feature control frame when the intent is to control a feature other than the pitch diameter of a threaded part.

planes, datum axes, and datum points. Standard symbols are used to identify datum features.

Datum Feature Symbol

The *datum feature symbol* consists of a square with a single capital letter or a rectangle with multiple capital letters, a leader line extending to the point of application, and a triangle at the end of the leader line. See [Figure 2-24](#). The *datum feature triangle* is typically shown filled. The datum feature symbol and triangle may be used to identify a surface or feature of size (either regular or irregular feature of size) as a datum feature. The given example only shows the symbols as they apply to surfaces. More information about application of datum feature symbols on surfaces and features of size is given in Chapter 6.

Datum letters are assigned alphabetically using single letters A through Z, omitting letters I, O, and Q. No two datum features on a single part should have the same datum letter. If all the letters of the alphabet are used and additional datum features are identified, a rectangle replaces the square and double letters are used for the additional datum features: AA–AZ then BA–BZ, etc.



Application to surfaces

Goodheart-Willcox Publisher

Figure 2-24. Datum feature symbols may be associated with the feature in any of several ways.

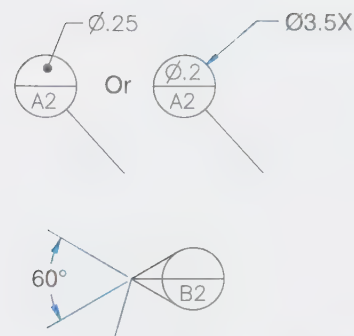
Application to Surfaces

Datum feature identification may be accomplished in a number of ways. In a CAD model, the datum feature symbol may be attached to the surface using a leader that terminates on the surface with a dot instead of an arrow. It is also a common practice to attach the datum feature triangle to the surface. This is best done with the triangle lying on a plane that is perpendicular to the surface. Also, in a CAD model, the datum feature triangle may be attached to a feature control frame that is applied to the surface.

In orthographic drawings, the datum feature triangle may be placed on an extension line from the surface. The side of the extension line to which the symbol is applied is not important. Another means for applying the symbol is to attach the triangle directly to the surface with a leader. The triangle may also be attached to a feature control frame that is applied to the feature.

Datum Target Symbol

Surface variation on a produced part or part size can make it impractical to use an entire feature to establish a datum. Datum targets may be identified on a part to require specific locations on the datum features to be used for establishing datum locations. See [Figure 2-25](#). Datum target symbols are used to identify each datum target. These symbols are not drawn at the target location, but are connected to target locations with a leader. No arrowhead is drawn on the datum target symbol leader.



Goodheart-Willcox Publisher

Figure 2-25. Datum targets must each be identified with a datum letter and number.

The current *datum target symbol*, implemented in 1982, is a circle with a horizontal line drawn across it. The datum identifying letter and a number are shown in the lower half of the circle. The letter identifies the datum. The number is the target number for that particular datum. Targets for each datum are numbered beginning with 1.

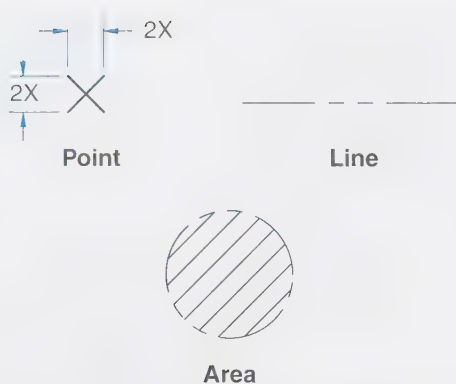
If datum target areas are used, the target size may be shown in the top half of the circle. Unless single-digit decimals are being used, the size value will not fit in the symbol. If two- or three-place decimals are being used, the size may be placed adjacent to the symbol with a leader extending from the size value to the top half of the symbol.

A *movable datum target* is indicated with a datum target symbol that includes two tangent lines forming a 60° angle and centered on the horizontal line through the symbol. The symbol may point to the right or left with the leader extending from the point.

Datum Targets

Datum target symbols may be attached to any of three datum target types. The target type shown on the part depends on many factors, including the design function and the characteristics of the part. The proper application of targets is explained in the chapter on datums.

Datum target points are indicated with two perpendicular lines, and the target point symbol is oriented so the lines of the symbol do not coincide with the object lines of the feature. See [Figure 2-26](#). The size of the symbol prevents it from being mistaken for a letter X. *Datum target lines* are indicated with a phantom line except when the target appears as an end view of the line. An end view of a target line is drawn with the same symbol as a datum target point. *Datum target areas* are outlined with a phantom line and filled with cross hatching. When a datum target area is small, it may be represented by a datum target point



Goodheart-Willcox Publisher

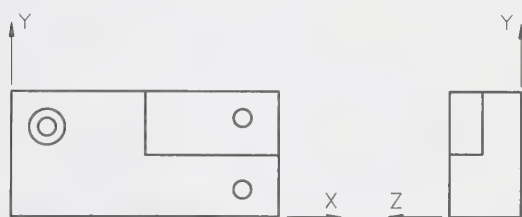
Figure 2-26. These are the three types of datum targets.

symbol, and the size of the area is specified inside or adjacent to the datum target symbol.

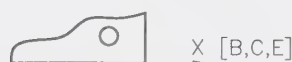
A **datum reference frame** symbol is used to indicate the three axes of a customized datum reference frame. See Figure 2-27. The symbol is made up of arrows that indicate the axis direction, and each of the three axes is labeled. The axes may be labeled in more than one view. When multiple datum reference frames exist, each of the axes may be noted to indicate the applicable datum reference frame. In the figure, the X axis is established by datum feature references to B primary, C secondary, and E tertiary. The axis is noted as X [B,C,E]. Uppercase letters are used to identify the axes.

Basic Dimensions

Every surface, every feature of size, and every feature location on a part is designed with an intended form, orientation, and location. Dimension values for these characteristics may be contained within the CAD data without being displayed in any view, or the dimension requirements may be



Datum reference frame identification



Labeling datum reference frame

Goodheart-Willcox Publisher

Figure 2-27. Datum reference frame symbol.

shown in the CAD model or on a drawing. Regardless of whether the dimensions are displayed or determined by querying a CAD model, it is not possible to build perfect parts to the “as designed” dimensions. Perfect fabrication processes do not exist; therefore, an amount of permissible variation must be defined for every characteristic of the part. The permissible variation is specified through the application of tolerances.

Permissible variation may be assigned by tolerances directly applied to a dimension, title block tolerances, a general note, or through the use of feature control frames. Geometric tolerances for orientation and location require that the orientation and location be defined by basic dimensions.

A **basic dimension** is a theoretically exact value for which there is normally a tolerance shown in a feature control frame. There is no directly applied tolerance on a basic dimension. A basic dimension is not an indication that a feature has zero tolerance. Wherever a basic dimension is seen, it is known that a feature control frame, or other control, identifies the amount of tolerance for the dimensioned feature.

Basic Dimension Symbol

A rectangle placed around a dimension identifies that dimension as basic. See Figure 2-28. Plus and minus tolerances do not apply to a basic dimension, and a feature control frame, or other control, is required to express tolerance for the dimensioned feature.

Noted Basic Dimensions

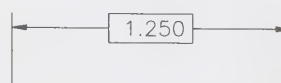
Drawing notes may be used to indicate that all untoleranced dimensions are basic. If all dimensions on the drawing are basic, the following general note may be used:

ALL DIMENSIONS ARE BASIC

Should it be necessary to use plus and minus tolerances on some of the size dimensions, the following general note may be used:

ALL UNTOLERANCED DIMENSIONS ARE BASIC

Title block tolerances do not apply to any dimension if either of these notes is used.



Goodheart-Willcox Publisher

Figure 2-28. A rectangle around a number identifies it as a basic dimension.

Notes in a CAD model may indicate that all CAD data is basic unless otherwise indicated. In this case, a hole with a size tolerance applied would have a size dimension and size tolerance as shown in the CAD model. Its size would not be considered basic. The hole would have a basic location and orientation.

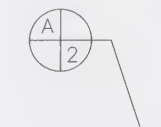
Past Practices

Drawings and CAD models may be created under contractual requirements from a customer. A contract may require compliance with a specific previous edition of the applicable standards, especially if the contract was created prior to publication of the current standard. This is common for long-term contracts, such as for aircraft. It is also common for parts that are in production and were designed when a previous version of the standards was in place. Many engines and transmissions in cars are in production over a long period of time, so the documentation for the parts may have been created in the past. As a result, the creation of drawings and CAD models under continuing contracts, and the ability to understand existing drawings, may require knowledge of previously used methods. Past practices related to some of the dimensioning and tolerancing symbols were noted in the preceding paragraphs of this chapter. The following section provides additional information about past practices.

Past Symbols

The symbology used for tolerance specification has gone through changes as the dimensioning and tolerancing practices have advanced to the current standard. Knowledge of past practices is needed for those situations in which an old drawing must be used or when a continuing contract still references an old version of the dimensioning and tolerancing standard.

Previous practice (prior to the 1982 standard) required the use of a circle divided into quadrants for the datum target symbol. See [Figure 2-29](#). The



1973 Practice

Goodheart-Willcox Publisher

Figure 2-29. Prior to 1982, the datum target symbol was divided into quadrants.

datum letter was placed in the upper-left quadrant, and the number was placed in the lower-right quadrant.

Prior to 1982, the current total runout symbol, with two arrowheads, was not used in the United States. See [Figure 2-30](#). The previously used total runout symbol looked like the symbol for circular runout, but the word TOTAL was shown beneath the feature control frame.

A separate symbol existed for symmetry tolerances prior to the 1982 standard. The 1982 version of the standard eliminated the symmetry symbol, and the position symbol was defined to be correct for specification of symmetrical location requirements. The symmetry symbol should not be used in documentation that must meet the 1982 standard requirements. The 1994 standard reinstated the symmetry symbol and clarified its definition.

A regardless-of-feature-size modifier was included in the standard prior to 1994. Effective in 1994 and continuing to be applicable today, tolerances are assumed to apply at RFS unless otherwise specified. Because tolerances are assumed to apply at RFS, the symbol is no longer needed. The 1994 standard included an alternate practice that allowed the application of the RFS symbol on position tolerances. The current standard does not support use of an alternate practice.

The datum feature symbol prior to 1994 was a rectangle. The rectangle included a datum identifying letter with a dash at each side of the letter.

Past Feature Control Frame Format

Feature control frames can look different depending on the standard that was in effect when the drawing was completed. One difference in appearance is caused by previously permitted options on the location of datum feature references

Past Practices			
Control	1994	1982	1973
Total runout			
Symmetry			
RFS			

Goodheart-Willcox Publisher

Figure 2-30. Some tolerance symbols have changed as a result of advancements in the dimensioning and tolerancing standard.

within the feature control frame. Another difference is that rules regarding the usage of material condition modifiers have changed.

Beginning with the 1982 standard, datum feature references are shown following the tolerance value. In accordance with the 1973 standard, datum feature references were permitted in either of two locations. They could be shown between the tolerance symbol and the tolerance value, or they could be shown in the same manner as defined for present practices. See **Figure 2-31**. Prior to 1973, datum feature references were required to be located between the tolerance symbol and tolerance value.

Prior to 1982, the diameter symbol was not required and the diameter abbreviation (DIA) was used. DIA followed the tolerance value.

A significant difference in the feature control frame appearance involves material condition modifiers. Prior to 1982, position tolerances were assumed to apply at the maximum material condition (MMC) if no modifiers were shown in the feature control frame. Assumptions were *not* permitted for positional tolerances in 1982. Beginning with the 1994 standard, the RFS modifier was assumed for all tolerances and datum feature references.

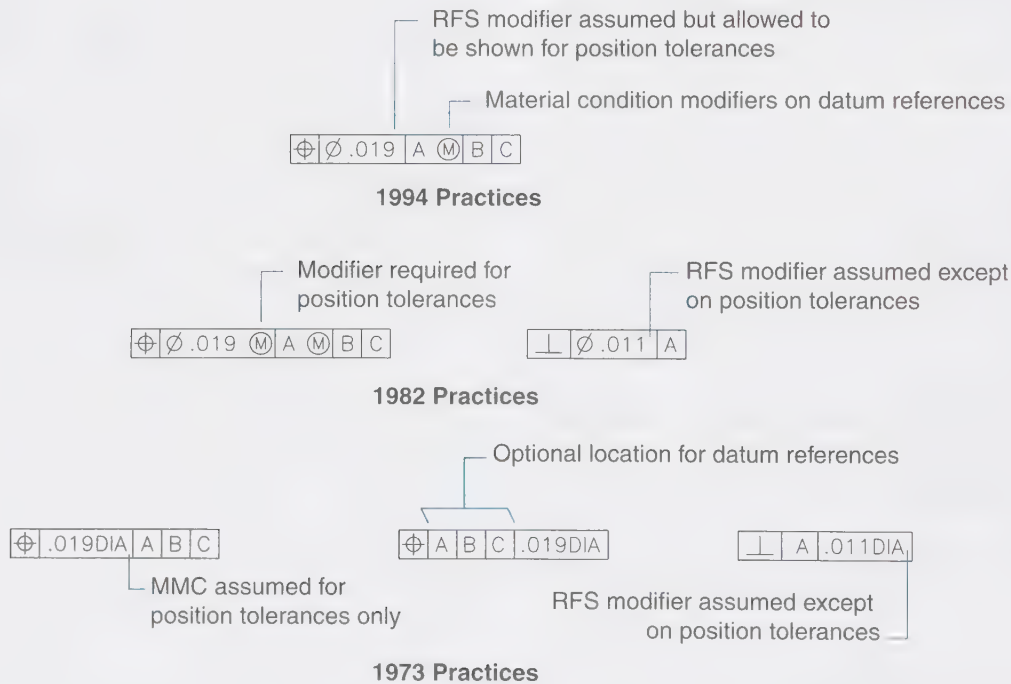
Past Basic Dimensions

Old drawings might include the word BASIC or the abbreviation BSC adjacent to some dimensions. These notations were permissible under early issues of the dimensioning and tolerancing standard. As part of the effort to replace words with symbology, the notations are no longer used.

Past practices used the abbreviation REF to indicate a dimension was for reference purposes. Current practice is to enclose the dimension value in parentheses.

Chapter Summary

- ✓ Symbology is now used for dimensioning and tolerancing of CAD models and drawings in place of the previously used notations and abbreviations. Symbology for general dimensioning and tolerance application is standardized and well defined.
- ✓ Dimensioning and tolerancing symbols, when correctly applied, provide better clarity of part requirements than can be achieved through the utilization of notes.
- ✓ There are standardized rules for applying dimensioning and tolerancing symbols. These rules must be followed properly if the full advantage of symbology is to be achieved.



Goodheart-Willcox Publisher

Figure 2-31. Previous standards included different feature control frame format requirements.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

- Symbols reduce the number of ____ used on a drawing.
 - views
 - words
 - dimensions
 - tolerances
- Symbols are ____ abbreviations.
 - preferred in place of
 - being gradually replaced by
 - not defined as well as
 - no longer used in place of
- The proportions for dimensioning symbols are ____.
 - based on a fixed character height of .125"
 - based on the character height used for dimensions
 - related to the character height of the drawing title
 - modified depending on the drawing scale
- A dimensioning symbol is usually placed ____ the number to which the symbol is applied.
 - under
 - over
 - after
 - before
- Straightness is a type of ____ tolerance.
 - form
 - orientation
 - position
 - angularity
- Perpendicularity is a type of ____ tolerance.
 - form
 - orientation
 - position
 - angularity
- The first compartment in a feature control frame ____.
 - must always show the tolerance symbol
 - sometimes includes a diameter symbol
 - shows either a tolerance symbol or datum feature reference
 - None of the above.

- A feature control frame may include a diameter symbol in the compartment with the ____.
 - tolerance value
 - datum feature references
 - tolerance symbol
 - None of the above.
- Position tolerances applied to threads control the location of the ____, unless noted otherwise.
 - mating part
 - pitch diameter
 - major diameter
 - minor diameter
- A datum feature is identified with a ____.
 - letter inside a square
 - number inside a rectangle
 - note
 - Either A or B.
- A dimension on a drawing may be made basic by ____.
 - drawing a circle around it
 - using a note
 - drawing a rectangle around it
 - All of the above.

True/False

- True or False?* The shapes of tolerancing symbols do not have any relationship to the indicated control.
- True or False?* The letter X with a space on each side of it means "by."
- True or False?* Flatness is a form tolerance.
- True or False?* Material condition symbols are not used in feature control frames.
- True or False?* Some feature control frames are not required to include datum feature references.
- True or False?* A dimension value may be identified as basic.

Fill in the Blank

- In a chamfer note showing .25 X .25, the symbol X is read as the word _____.
- How many orientation tolerance symbols are there? _____
- Form and orientation tolerances are specified in a _____.

21. A rectangle drawn around a dimension value indicates that the dimension is _____.
22. The abbreviation for maximum material condition is _____.

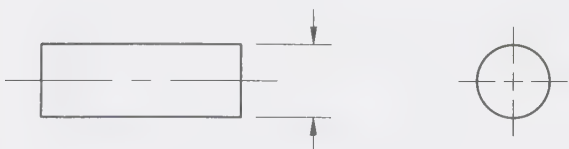
Short Answer

23. Describe what the letter **X** means when no space is placed between it and a preceding number.
24. List the form tolerances.
25. Sketch each of the following tolerance symbols.
 - A. Straightness
 - B. Perpendicularity
 - C. Total runout
 - D. Line profile
 - E. Position
26. Sketch each of the following dimensioning symbols.
 - A. Diameter
 - B. Counterbore
 - C. Spotface
 - D. Countersink
 - E. Slope
27. Based on the current standard, in which location(s) are the datum feature references placed in a feature control frame?
28. Sketch the two material condition modifier symbols that may be shown following a tolerance value in a feature control frame and label each of them.

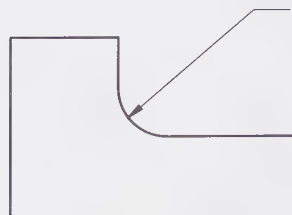
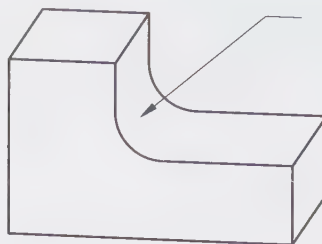
Application Problems

Some of the following problems require that a sketch be made. All sketches should be neat and accurate. Each problem description requires the addition of some dimensions for completion of the problem. Apply all required dimensions in compliance with dimensioning and tolerancing requirements.

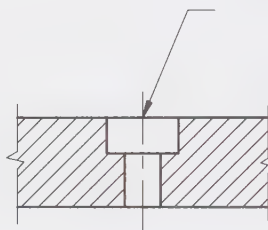
29. Show a 1.125" diameter for the given shaft.



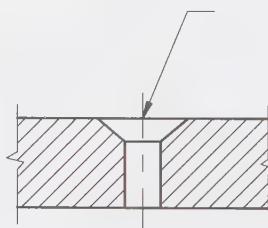
30. Show a .625" radius on the inside corner.



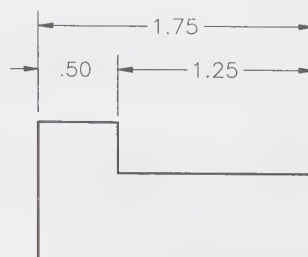
31. Use symbology to specify the counterbore and hole size in a note attached with a leader. The hole has a .282" diameter. The counterbore is .438" diameter and is .375" deep.



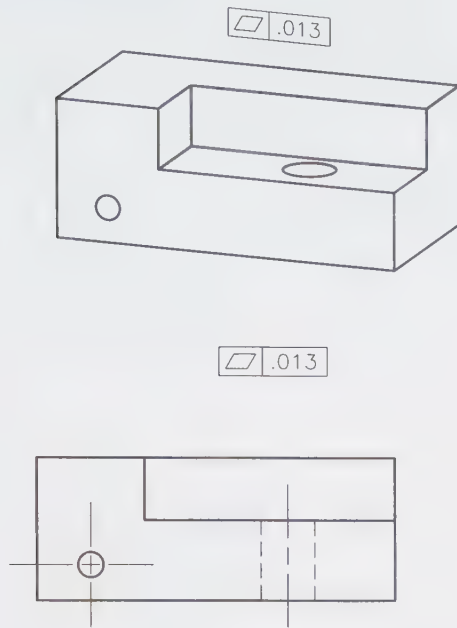
32. Use symbology to specify the countersink and hole size in a note. The hole has a .218" diameter. The countersink has an 82° angle and a .438" diameter.



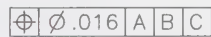
33. Make the 1.25" dimension a reference value.



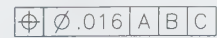
34. The given flatness tolerance specification is to be drawn so that it applies to only the top surface of the given part.



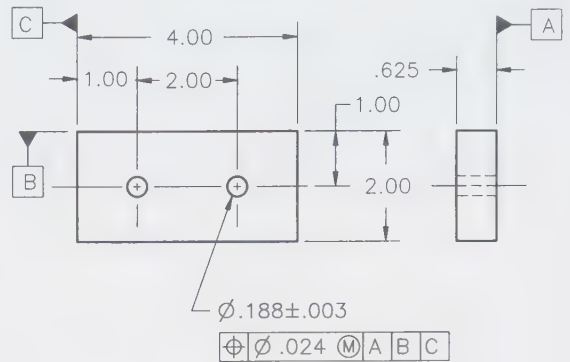
35. In accordance with the 1994 standard, the given tolerance value is applicable at what material condition?

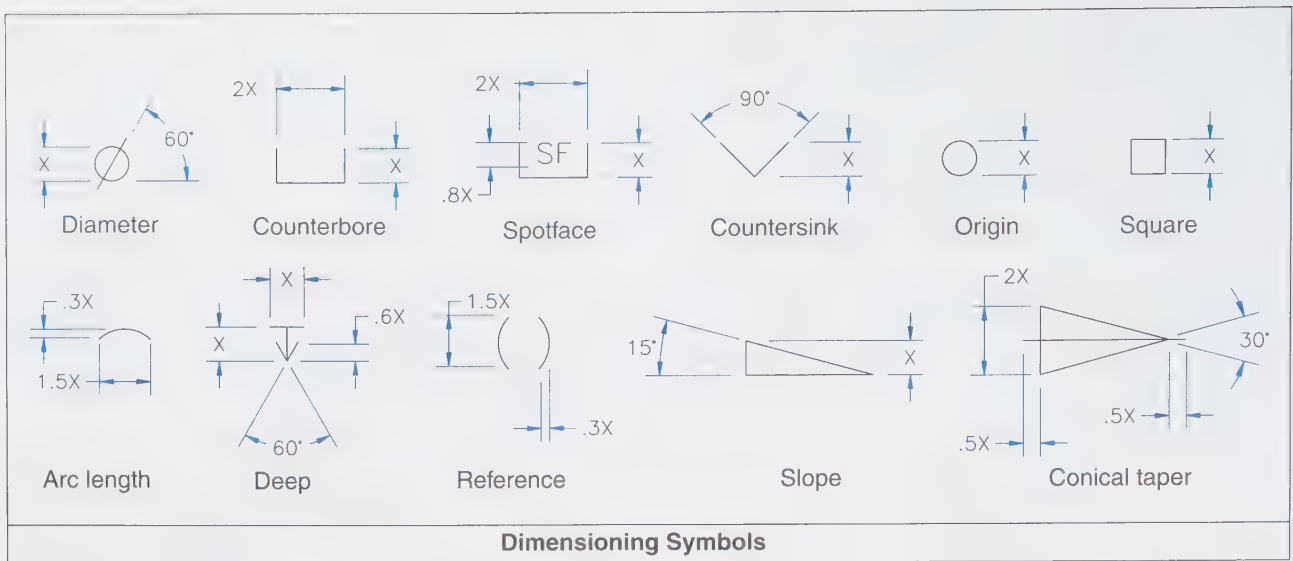


36. In accordance with the current standard, the given tolerance value is applicable at what material condition?

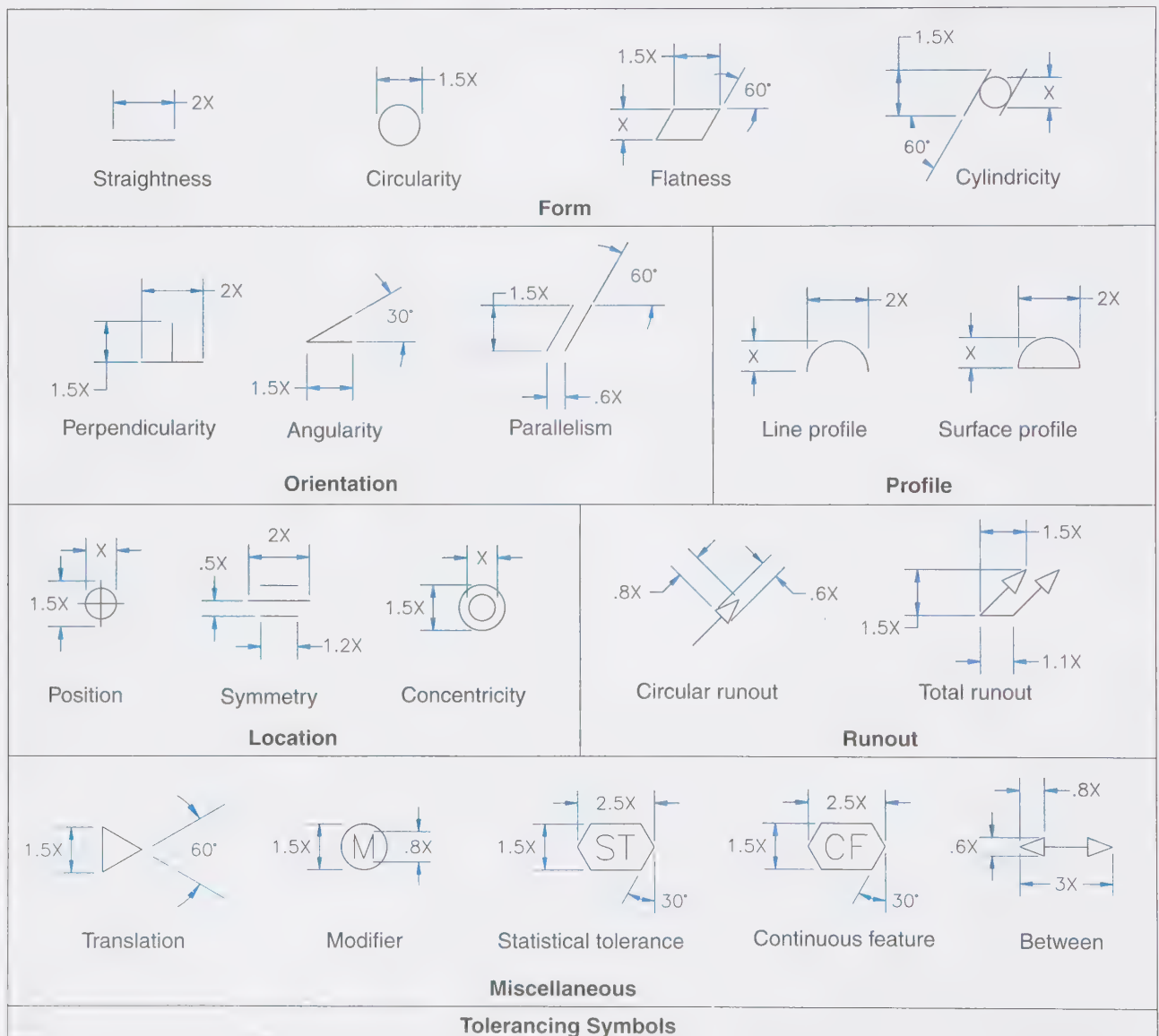


37. Make all hole location dimensions basic.





Goodheart-Willcox Publisher



Goodheart-Willcox Publisher

Chapter 3

General Dimensioning Requirements

Objectives

Information in this chapter will enable you to:

- ▼ Apply general dimensioning methods using the correct line types, lettering sizes, and arrowhead form.
- ▼ Describe and apply general dimensioning systems including chain, baseline, rectangular coordinate, and polar coordinate dimensions.
- ▼ Utilize preferred dimension placement to provide clear part requirements specification.
- ▼ Apply general and specific notes on a drawing.
- ▼ Cite the general categories of fit between mating parts.

Technical Terms

aligned dimensioning
arrowheads
baseline dimensioning
basic dimension
chain dimensioning
clearance fit
cover sheet
decimal inches
dimension lines
dimension origin
double dimensioning
extension lines
feature control frame
hole identification symbol
interference fit
joggle
leader lines
leaders
least material condition (LMC)
line fit
location dimension

material boundary modifiers
material condition modifiers
maximum material condition (MMC)
note sheets
parts list sheets
polar coordinate dimensioning system
rectangular coordinate dimensioning system
reference dimension
regardless of feature size (RFS)
SI (metric) system
size dimension
station points
title block tolerances
tolerance accumulation
transition fit
unidirectional dimensions
US customary (inch) system
zero allowance

Introduction

The practices described in this chapter were established for dimensioning orthographic drawings. These practices may also be utilized for orthographic drawings created from three-dimensional computer-aided design (CAD) models. These practices are not always applicable in three-dimensional models because some companies query the CAD geometry to resolve dimensional requirements. *Querying* means to select one or more features to obtain information. When queried, the software resolves the needed information by performing the necessary calculations to display the requested information. When CAD geometry is accurately modeled, it is possible for downstream users, such as those in manufacturing and inspection, to query the geometry and obtain dimensional information.

Dimensions applied to orthographic and pictorial drawings provide the sizes and locations of features. Proper application of dimensions makes size and location requirements easier to understand and increases the probability that the parts will be properly produced. Dimension application requirements are defined by a standard of the American Society of Mechanical Engineers (ASME). The standard is identified by the following designation: ASME Y14.5-2009.

Proper application of the size and location dimensions in orthographic and pictorial drawings, supplemented by tolerance specifications as defined in the following chapters, provides complete dimensional control for a part. Dimensions must be shown on any drawing that is to be used for the production of parts. This is true regardless of whether the drawing is produced by manual means or by a CAD system.

It is possible that a CAD-generated design will be fabricated directly from the design file data without the generation of an orthographic or pictorial drawing. If a CAD design file is used without the generation of a dimensioned drawing, some dimensions defining locations, sizes, and tolerances may still need to be added to the file. Although all entities in the CAD design file have dimensional data associated with them, this data does not typically indicate allowable variations. The allowable variations are therefore defined through the application of tolerances.

As CAD systems become more capable, the file entities may have attributes assigned to them to indicate allowable tolerances. However, the methods for identifying allowable variations as attributes on design file entities have not been standardized at this time. The display requirements for tolerances in CAD models are established, but assigning tolerances as attributes on CAD features for use in downstream software (such as machining or inspection systems) is not yet standardized or fully developed.

Although tolerances in a CAD model are not always readable by downstream software, the current capabilities of CAD systems have, to some extent, reduced the need to show the as-designed (nominal) dimensions. The as-designed dimension values in the CAD models are obtainable and can be queried (interrogated) by the CAD user without any need to apply dimensions in a view. To minimize time spent querying models, some companies still create dimensioned views that are in compliance with the guidelines in this chapter.

Dimensioning Methods

Standard dimensioning methods are used to provide maximum clarity and consistency in the presentation of product requirements. Dimensioning systems are universally understood throughout the United States because national standards define dimensioning and tolerancing methods. ASME Y14.5 defines the meaning for dimensions and tolerances and their application on orthographic drawings, and to some extent the application on CAD models. ASME Y14.41 defines some of the tolerance application methods for CAD models. ASME Y14.41 was developed to define 3D tolerance application requirements without changing the tolerance meanings defined in ASME Y14.5.

Compliance with standards is necessary so information can be easily interchanged throughout the United States and internationally. When companies depart from national standards, it can cause confusion.

It is impossible to write a single document that addresses every possible dimensioning situation. Therefore, the national standards address common situations and requirements through a set of principles that can be extended to most applications. Because there may be an occasional special situation that cannot be addressed by extension of the standard principles, individual companies may establish their own drafting and dimensioning requirements for special applications. Care should be taken to create company specific standards that do not conflict with the national standards.

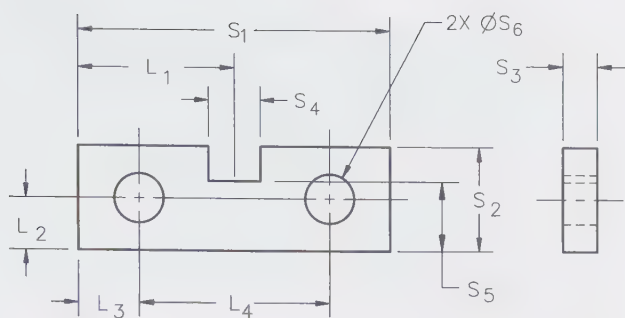
Standards Advisory

ISO Standards

It is important to be aware of the existence of ISO standards that define dimensioning and tolerancing requirements. ISO standards are similar to ASME standards and use many of the same symbols. However, ISO standards have modified the meaning of some symbols in a way that can potentially result in significant differences in interpretation. The ASME standards are well understood internationally, and compliance with ASME standards should be much easier to achieve.

Location and Size Dimensions

All dimensions applied to a part show size, angle, or location. See **Figure 3-1**. A *location*



Goodheart-Willcox Publisher

Figure 3-1. The letter S indicates size dimensions, and the letter L indicates location dimensions.

dimension (specified with an L in the figure) describes *where* a feature is, and a **size dimension** (specified with an S in the figure) indicates how large a feature is. Location and size must be controlled in all three axes.

Most feature locations are specified with one or two dimensions. This is possible because the remaining location dimension is defined by other features on the part when only two dimensions are given. The location of a drilled hole is an example; two coordinates locate its center, while the surface into which it is drilled locates the start of the hole.

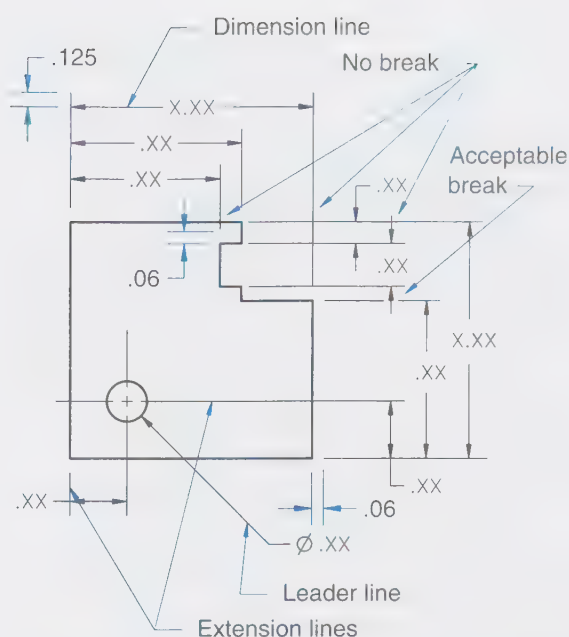
Size is dimensioned through one of several means depending on the shape. A rectangular prism is sized by dimensioning the height, width, and depth (three size dimensions). A hole can be sized by giving its diameter and depth. If a hole goes through a part, its size is specified by giving only the diameter. The part thickness defines the length of a through hole.

Line Use

The three line types used for dimensioning are extension lines, dimension lines, and leader lines. All of them are drawn with thin lines. When two line widths are used in drawings, the thin lines provide enough contrast with the wide object lines to avoid confusion between the object outline and the dimensions.

Extension Lines

Extension lines are used to extend features on an object to allow the application of dimensions. See **Figure 3-2**. ASME Y14.5 requires that a visible gap exist between the object and extension line. It is generally a good practice to begin an extension line approximately .031" to .062" (0.75 to 1.5 mm) from the feature. When showing dimensions on a CAD model, the extension line gap is not required. Extension lines extend approximately .125" (3 mm)



Goodheart-Willcox Publisher

Figure 3-2. Extension lines are used to extend features for the application of dimensions.

beyond the last dimension line. CAD systems often have these predetermined distances as system defaults, but many systems also permit adjustment of the preset values.

Extension lines may cross object lines or other extension lines. No breaks are made in an extension line when it crosses an object line or another extension line. Breaks in extension lines cause a discontinuity that makes the drawing harder to read. It is only acceptable to break an extension line if it crosses or is sufficiently close to an arrowhead as to cause confusion.

Extension lines are usually perpendicular to the dimension lines to which they extend, and in general practice are perpendicular to the dimension line defining the size or location of that feature. An extension line from the center of a circular feature starts adjacent to the centerline cross and extends to the dimension line. CAD limitations may require that the extension line start at the center point.

Sometimes, there is a need for a special treatment of extension lines. Special treatments include radial extension lines, oblique extension lines, and an offset (joggle) inserted in an extension line, as explained later in this chapter.

Dimension Lines

Dimension lines show the direction and magnitude of a dimension. The direction of a dimension line is parallel to the distance being specified. The dimension magnitude is shown by the value

inserted in the dimension line. Arrowheads are placed at each end of a dimension line to show the point of application.

There are four arrangements in which the dimension line and values can be shown. See **Figure 3-3**. The preferred arrangement is with the arrowheads and value inside (between) the extension lines. This arrangement is used when sufficient space exists between the extension lines to show the arrowheads, dimension line, and value.

When the distance between extension lines is limited, one of the following arrangements is used:

- The arrowheads may be placed inside and the value outside.
- The value may be placed inside and the arrowheads outside.
- Both the arrowheads and value may be placed outside.

When a combination of dimension line and value arrangements is used, it is acceptable for one arrowhead to be used for two dimensions as shown in **Figure 3-3**.

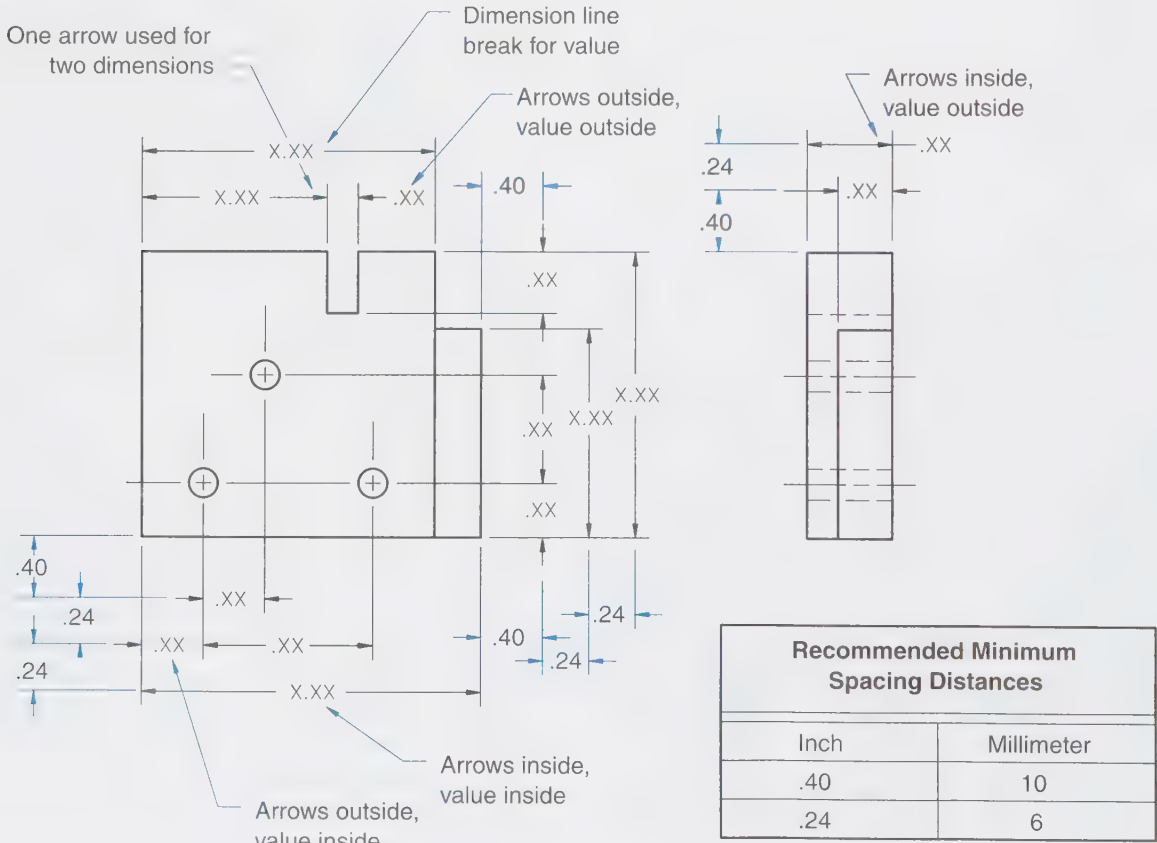
Dimension lines must be adequately spaced to make them easy to read. The minimum recommended space between the outline of the part

and the first dimension line is .40" (10 mm). The minimum recommended distance between succeeding dimension lines is .24" (6 mm). Preferable spacing is .50" (13 mm) from the object and .38" (9 mm) between succeeding dimension lines. These distances are general guidelines; variation is permitted, provided the dimensions do not run together when reduced-size copies of the drawing are made. The spacing used between dimension lines should appear constant to provide an easy-to-read drawing.

The distance to the first dimension line is measured from the outline of the object. If there is an offset in the part outline, it is preferable to place the first dimension outside the outline of any adjacent features. See **Figure 3-4**. A remotely located feature such as shown in the given figure does not affect the location of the first dimension. It is a matter of judgment as to when a feature becomes remotely located.

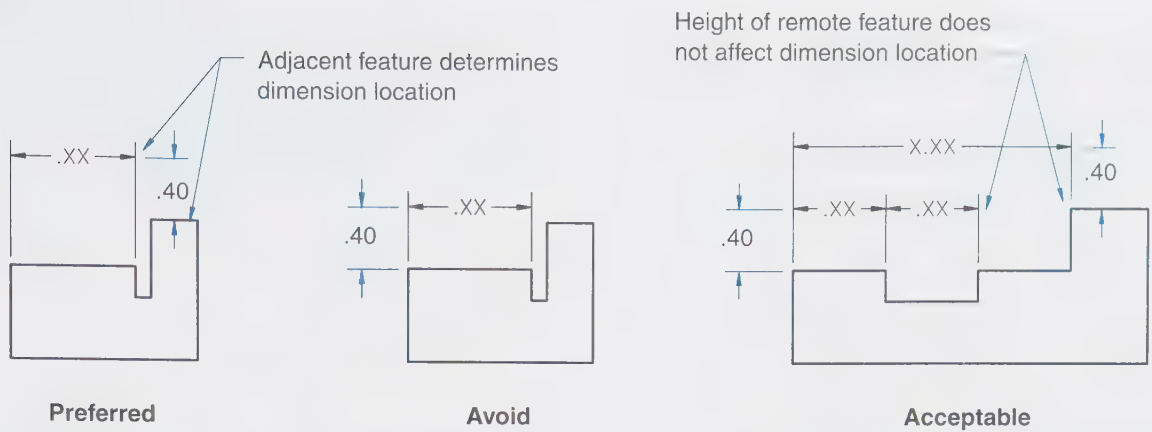
Leader Lines

Leader lines, or leaders, are used to connect information, such as a note or symbol, to a specific feature on the part. See **Figure 3-5**. The leader line



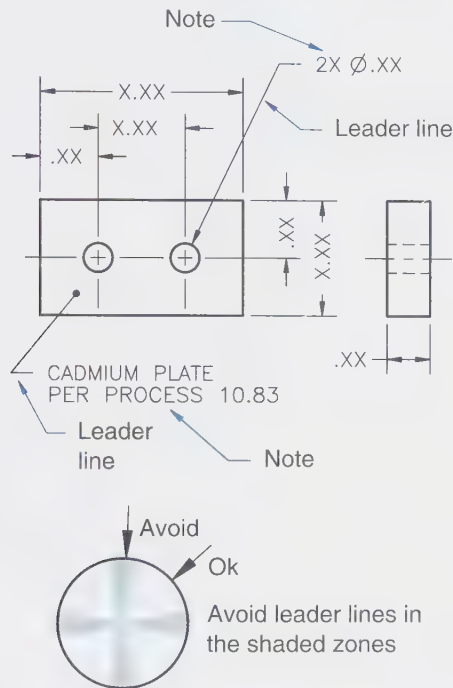
Goodheart-Willcox Publisher

Figure 3-3. Proper dimension line spacing allows each dimension to be seen clearly. The minimum recommended spacing is shown.



Goodheart-Willcox Publisher

Figure 3-4. Adjacent features must be considered when locating dimensions.



Goodheart-Willcox Publisher

Figure 3-5. Leader lines are used to connect notes and drafting symbols to specific features on a drawing.

extends from the first or last character in the note. A short horizontal line normally extends from the first or last character, and a leader extends from the horizontal line to the noted feature. Some companies omit the horizontal line. The leader is terminated with an arrowhead or dot. An arrowhead is used if the leader terminates on the profile of the feature, and a dot is used if the leader terminates on a surface.

The horizontal bar on the leader is drawn with a visible gap between it and the character. The size of an arrowhead is proportional to the line width. Dot terminators are .062" (1.5 mm) in diameter.

Leader lines are not drawn horizontally or vertically; they are always inclined. Horizontal or vertical leaders could be confused with dimension and extension lines. Leaders may cross object and extension lines and are not broken at the intersections. Intersections between leader lines and dimension lines are to be avoided. A leader line should be broken if it crosses an arrowhead.

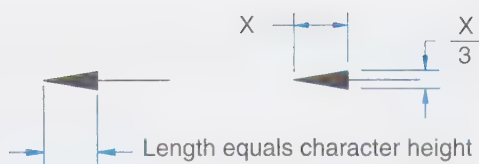
Arrowheads

Arrowheads are used on dimension lines and leader lines to show the point of application. Arrowhead size is proportional to the line width (ASME Y14.2-2008). However, a common practice that results in an acceptable size is to make the arrowheads equal in length to the character size (height) used for dimensioning and notes. If .12" characters are used, .12" long arrowheads are used. Arrowheads are made with a length-to-width ratio of 3:1. A .12" long arrowhead is .04" wide. See [Figure 3-6](#).

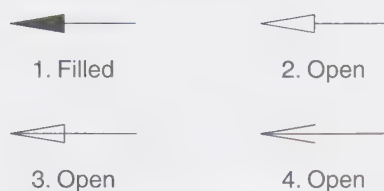
All arrowheads shown in ASME Y14.5 are filled except where optional arrowhead styles are explained as being acceptable. ASME Y14.2-2008 illustrates four acceptable arrowhead styles. All four styles are acceptable with the order of preference shown in the figure. The use of solid arrowheads is recommended.

Character Size and Style

The height of all numbers and letters used for the dimensions and notes on a drawing must be the same. The minimum character size for all drawing sizes is standardized at 3 mm (.120") for dimensions and notes. The minimum character size for cutting plane letters is 6 mm (.24"). Considering the current quality of fonts and printing, these character sizes are allowed.



Arrowhead proportions



Arrowhead forms

Goodheart-Willcox Publisher

Figure 3-6. Arrowheads may be open or filled. The order of preference is as numbered.

Past practice established character size on the basis of drawing sheet size. The larger character size was intended to ensure that hand drawn letters would clearly print. Drawing sheets that were 17" × 22" or smaller required a minimum character height of .125". Drawing sheets larger than 17" × 22" required a minimum character height of .156". The character size for large sheet sizes permitted clear reproduction of the characters on reduced size copies.

All characters on a drawing are to be of one style. Characters may be vertical or inclined with a recommended angle of 68° from horizontal for inclined characters. When an existing drawing is revised, the character style used for the revision should match the character style already on the drawing.

Dimension values are centered on the dimension line. See **Figure 3-7**. A break in the dimension line is made at the dimension value when the dimension value is placed between the extension

lines. The break is large enough to leave a visible gap between the dimension line and number.

Unidirectional Dimensions

Unidirectional dimensions show all dimension values written horizontally so they are readable from the bottom of the page. Unidirectional dimensions are used exclusively in ASME Y14.5 when applied to orthographic views.

Unidirectional dimensions do require more space between vertical dimension lines than is normally required for horizontal dimensions. The increased space for vertical dimensions is required to prevent dimension values from overlapping or crowding adjacent dimension lines.

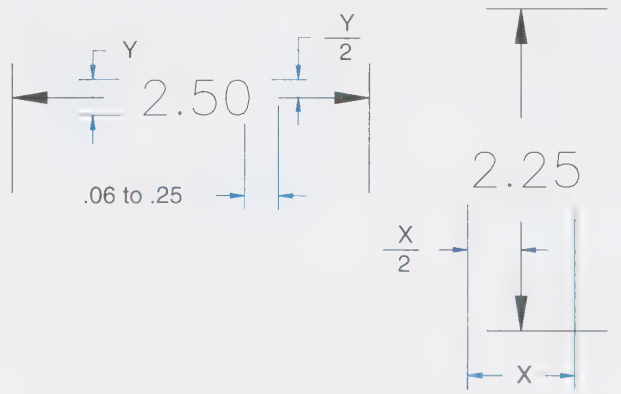
All values are applied in the same orientation, regardless of the dimension line orientation. See **Figure 3-8**. Increased dimension line spacing for vertical and inclined dimensions requires an increased amount of area for a drawing.

Aligned Dimensions

Dimension values are parallel to the dimension lines when the *aligned dimensioning* method is used. See **Figure 3-9**. Horizontal dimensions are written from left to right, and read from the bottom of the page. Vertical dimensions are written from bottom to top, and read from the right side of the page. Aligned dimensions are no longer illustrated in the dimensioning and tolerancing standard, but they continue to be used by some companies. The use of aligned dimensions is not recommended on orthographic views.

Aligned dimensions must be read from the bottom or right side of the drawing sheet when applied on multiview drawings. The orientation of some dimension lines can make it difficult to achieve the required orientation of the dimension text. See **Figure 3-9**. When an undesirable orientation of dimension lines is required, the dimension values may be rotated to the horizontal orientation. Any dimension entered in this way does not conform to the aligned dimensioning method, but is preferable to a value that is read from the left side or top of a drawing.

The unidirectional and aligned dimensioning methods are not to be mixed within a drawing. One method or the other must be selected. Normally, only one method is used on drawings within a company. When dimensions are applied to features on a CAD model without drawing views, the dimension value is aligned with the dimension line.



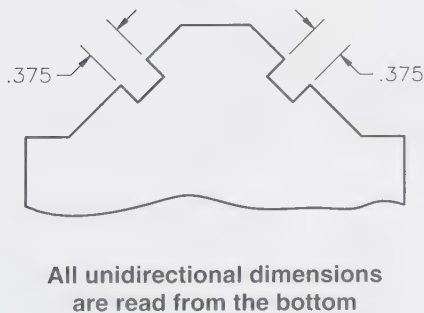
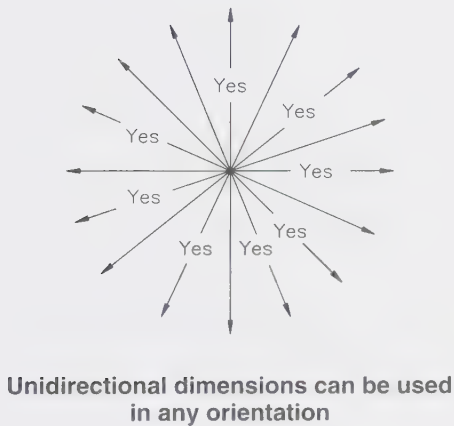
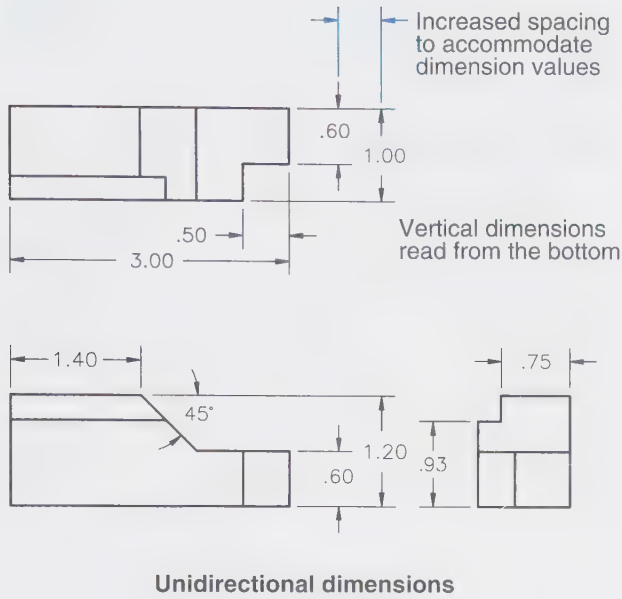
Goodheart-Willcox Publisher

Figure 3-7. Dimension values are centered on the dimension lines.

Scale

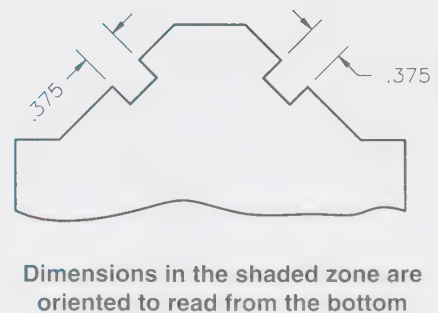
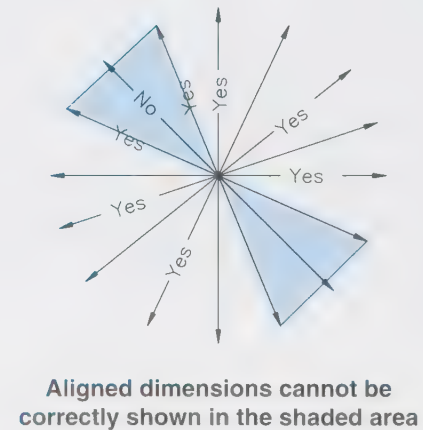
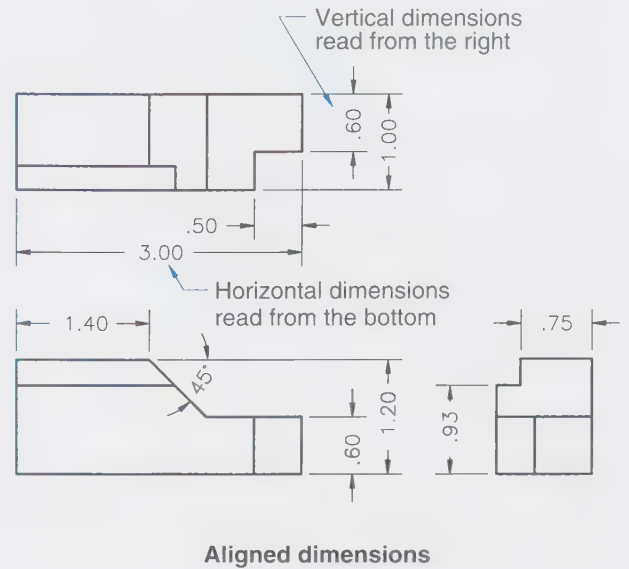
All views in orthographic and pictorial mechanical drawings are drawn to scale. Regardless of the scale at which views are drawn, the

dimension values are those to which the part is to be made. See [Figure 3-10](#). An 8" feature drawn at half scale is shown 4" long, but the dimension must indicate the 8" requirement. Model geometry in a CAD system should be full scale.



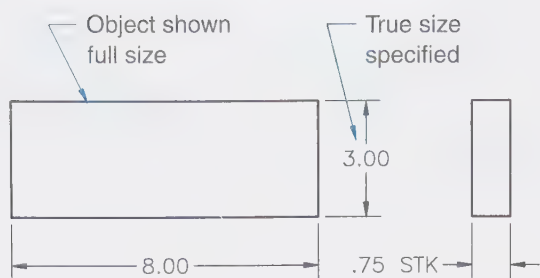
Goodheart-Willcox Publisher

Figure 3-8. All dimensions are read from the bottom of the page when using unidirectional dimensions.

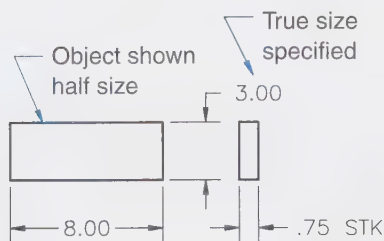


Goodheart-Willcox Publisher

Figure 3-9. Aligned dimensions cause difficulty in application for some orientations.



SCALE = 1/1



SCALE = 1/2

Goodheart-Willcox Publisher

Figure 3-10. The part size to be produced is shown in the dimension regardless of the drawing scale.

The scale of a drawing is always noted in the title block. Scale may be noted in fractional form, such as 2/1, 1/1, 1/2, and 1/4. The scale notation is not provided to make it possible to scale the print to determine dimensions; rather, it is noted to give the reader a quick reference for mentally forming a true-size image of the part. The noted scale also informs everyone of the scale to use if the drawing original must be revised.

A reproduced copy of a drawing should never be scaled (measured) to determine the size of a feature. Scaling a reproduced copy is a risky practice. First, if a dimension is missing from the drawing, the drawing is incomplete. An incomplete drawing is proof that a mistake has been made, and it is possible that more mistakes exist. The location or size of the undimensioned feature may be drawn incorrectly. Second, the missing dimension may be a critical value that was calculated and then accidentally omitted from the drawing. If the dimension is critical, the gross measurement from a copy of the drawing will not reflect the accuracy needed. A third reason for not scaling a drawing is the imperfect process of making a reproduced copy. Most reproduction processes cause either a slight reduction or enlargement in at least one direction along the copy. Inaccurate reproduction processes guarantee that any measurements made from the drawing are incorrect.

Dimensions are not required when using a solid model to define fabrication dimensions. Dimension values are determined by querying the model. This is becoming a common practice when the designing organization and fabrication organization have compatible CAD and production software. Tolerances and datums are usually shown regardless of whether dimensions are applied to the model.

Measurement System

The two measurement systems used today are the *US customary (inch) system* and the *SI (metric) system*. The inch system is used extensively in the United States, but the metric system is becoming more widely accepted. ASME Y14.5 is illustrated using metric values. Some US-based international companies, such as those in the automotive industry, are mostly metric. Others, such as the aircraft industry, are mostly inch based. Most countries outside the United States use the metric system.

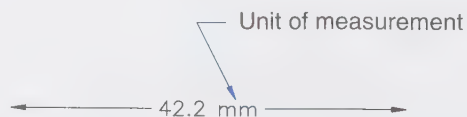
The application of dimensions is affected very little by which measurement system is used. Only the representation of the dimension value is different.

The measurement system used on a drawing should be noted in or near the title block. This is especially true of drawings completed using the metric system. If no drawings are created on sheets with a title block, then a note should be entered in the CAD model to identify the measurement system. Whether used in a drawing or in a CAD model, the note should state:

UNLESS OTHERWISE SPECIFIED,
ALL DIMENSIONS ARE IN MILLIMETERS
or

UNLESS OTHERWISE SPECIFIED,
ALL DIMENSIONS ARE IN INCHES

The noted measurement system is used for all dimensions. Any exceptions are noted by showing the unit of measurement beside the dimension value. See **Figure 3-11**. When the measurement system noted in the title block is inches, any metric



Goodheart-Willcox Publisher

Figure 3-11. The unit of measurement is shown on dimensions that are different from the predominant measurement system used on the drawing.

dimension shown on the drawing is followed by the abbreviation for millimeters (mm). There is no period following the abbreviation for millimeters. When the noted measurement system is millimeters, any inch dimension shown on the drawing is followed by the abbreviation for inches. Generally, the abbreviation for inch is IN without a period following it. Although the ASME Y14.5 standards in 1982 and 1994 showed the abbreviation with a period, it was commonly used without a period. The unit of measurement is not shown on dimensions that are based on the measurement system noted in the title block.

Regardless of which unit of measurement is being used, it is sometimes desirable to show both the inch and metric values. When both values are shown, one of them must be a reference value. If the inch value is the firm requirement, the metric dimension is placed in parentheses to indicate that it is reference information. See **Figure 3-12**. The reference value is placed directly below the firm dimension requirement, and both values are placed in the same break in the dimension line. A note such as the following is added to the drawing that states:

VALUES SHOWN IN PARENTHESES
ARE METRIC AND FOR REFERENCE ONLY.

This note makes it clear that all production and inspection measurements are to the firm dimension values.

The same methods may be used on metric drawings to show a reference inch value. The metric value is shown as a firm requirement, and the inch dimension is shown as a reference value in parentheses. The reference inch value is placed below the metric value. A note such as the following is added to the drawing that states:

VALUES SHOWN IN PARENTHESES ARE
IN INCHES AND ARE FOR REFERENCE ONLY.

Conversion of dimension values from one system to the other must be approached with a great

deal of caution. The two values must be equal. This not only includes the as-designed “nominal” value, but also any tolerance applied to the as-designed value. Conversion from one measurement system to the other requires the use of an accurate conversion factor. Rounding off converted values must be avoided until the final calculation is completed. Rounded-off values must not be used in the calculation of other values. The accumulation of round-off errors can cause unacceptable changes in part dimensions. Accurate conversion from one measurement system to the other should be based on 25.40 millimeters per inch. Inches are converted to millimeters by multiplying the inch value by 25.40 millimeters per inch. For example:

$$.875 \text{ inch} \times 25.40 \text{ mm/inch} = 22.23 \text{ mm}$$

Inch Measurement System

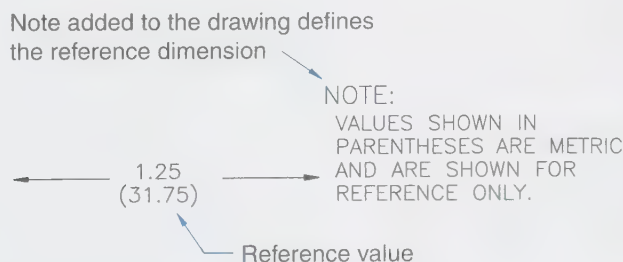
Dimension values shown in inches should be shown as *decimal inches*. Decimal inch values may be displayed as whole numbers or in tenths, hundredths, thousandths, or even smaller increments, depending on design requirements.

History Brief

Fractional Dimensions

The application of fractional dimensions is not defined by the current dimensioning standard. Fractions were widely used in the past and some old drawings may contain them. Fractions should not be used on new engineering drawings that are intended to comply with ASME Y14.5-2009.

Applications in today's industry often require that a high degree of accuracy be maintained in the manufacture of products. The required degree of accuracy makes fractions impractical. As an example, installation of some bearings requires that shaft sizes be produced to within .0005" (five ten-thousandths)



Goodheart-Willcox Publisher

Figure 3-12. A reference value may be shown in a unit of measurement different from the firm requirement. When both measurement system values are shown for one dimension, one value is identified as reference.

of diameter. Specifying dimensional control of this magnitude does not permit the use of fractions. The use of decimal dimensions permits specification of allowable part variation to any necessary degree of accuracy. Of course, the specified tolerance should never be smaller than what is needed to achieve the intended design function.

The decimal system provides a system in which mathematical operations are easier to perform. Adding, subtracting, multiplying, and dividing operations are all easier to perform on decimal numbers than on fractions.

Total freedom in size increment is achieved by using decimal measurements. Some designers attempt to maintain fractional or .10" size increments, even when using the decimal system. The practice is not necessary except when standard sizes for vendor-supplied items, such as screws, seals, and bearings, are based on nominal fraction sizes.

A zero is not placed in front of a decimal inch measurement less than one inch. The number of decimal places normally shown is two or three; but one, four, or more may be used. The number of decimal places does not affect the accuracy of the specified value. The decimal values .23, .230, and .23000 are exactly the same.

There is a common practice of specifying **title block tolerances** that apply according to the number of decimal places shown in the dimension value. This is typically done when dimensions are applied in a drawing and not done when an undimensioned CAD model is used. Tolerances are shown as in the following example:

.XX	±.02
.XXX	±.010
ANGLES	±.5°

The required accuracy for producing a .23" dimension is ±.02" because of the tolerances shown in the example. The required accuracy for producing a .230" dimension is ±.010". The required accuracy for producing a .2300" dimension is unknown because the title block does not show a tolerance for a four-place decimal. A tolerance for four-place decimals could be added to the title block, or a specific tolerance could be applied directly to the dimension.

Proper use of decimal values requires an understanding of how to *round off* numbers. When using two- or three-place decimal equivalents for common fractions, some of the decimal values must be rounded off. An example is a feature that is 3/32" in size. The decimal equivalent of 3/32" is .09375". The two-place decimal equivalent is .09", and the three-place decimal equivalent is .094".

When a number is to be rounded off, specific rules are to be followed. The rules used to round off decimal values are:

- If the value following the last figure to be retained is less than 5, do not change the last figure. (Rounding to two places, .3125 rounds off to .31, because the figure following the 1 is less than 5.)
- If the value following the last figure to be retained is greater than 5, the last figure is increased by 1. (Rounding to two places, 2.1261 rounds off to 2.13, because the figure following the 2 is greater than 5.)
- When the value following the last figure to be retained is exactly 5, the last significant figure is made an even value. Rounding to three places:
.31250 rounds off to .312
.43750 rounds off to .438

Rounding off values is necessary when using two- and three-place decimals for dimensioning. When a number is rounded off, it is no longer an exact equivalent of the fraction value. The allowable tolerance is applied to the rounded-off value, not to the fractional value.

A 3/32" value rounded to two places becomes .09" and is the same as .090000". Once the rounding is done, specifying the rounded dimension as .09 or .090 only affects which of the title block tolerances is applicable.

Part of the US customary measurement system is the foot. In machine drafting, dimensions under 72" are shown in inches. No symbols are used to indicate the unit of measurement when a dimension only shows inches. Some companies show dimensions greater than 72" in feet and inches. Symbols are used when both feet and inches are shown. A dash is used to separate the number of feet from the number of inches.

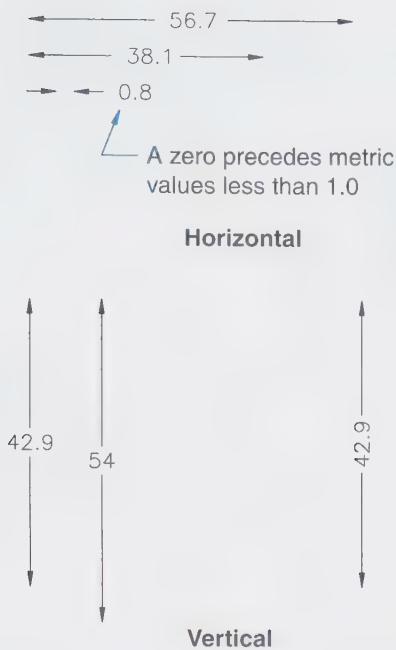
Example:

9'-3.50" or 111.50"

Metric Measurement System

The standard unit of measurement for drawings using the metric measurement system is the millimeter. One thousand millimeters equals one meter, and 25.4 millimeters equals one inch.

Metric values are entered in dimensions the same way as decimal inch values. See [Figure 3-13](#). Either the aligned or unidirectional method of dimensioning may be used. All metric dimensions are shown using decimals. One-place decimals are common. A zero is placed in front of the decimal point for values smaller than one. This is different



Goodheart-Willcox Publisher

Figure 3-13. Decimal divisions are used for metric measurements.

from the inch system, where a leading zero is not shown. A trailing zero is not shown for whole millimeter values unless limit dimensioning is used, in which case all values show the same number of decimal places. The figure shows a 54 mm dimension that has no trailing zero. The abbreviation for millimeters (mm) is not shown except when a metric dimension is applied on a drawing that is predominantly based on the inch system.

Dimension Systems

Several systems exist for the application of dimensions. Each system has specific applications for which it is best suited. The systems are often used in combination to achieve maximum control over the size of a part.

Chain, rectangular coordinate, polar coordinate, and direct (combined) dimensioning systems are used to show size and locations. These systems define the manner in which dimensions are applied and if the dimensions are directly toleranced will affect the location tolerances for features. If several features are dimensioned from one line, each feature's location varies in relationship to that line. If features are located by a chain of dimensions, the location tolerance for each feature in the chain is added together. The tolerance effects of the dimensioning system may be overcome by using basic dimensions and geometric tolerances as explained in following chapters.

Pro Tip

Plus and Minus Location Tolerance Risk

When location dimensions have plus and minus tolerances on them, the tolerances are subject to interpretation errors. Surface form and orientation variations raise questions about the correct origin and the direction for measurements. Plus and minus location tolerances are shown in the ASME Y14.5 standard on a very limited number of examples, and those examples do not establish guidelines for interpretation when surface form and orientation tolerances are present. For this reason, plus and minus tolerances for location dimensions are not recommended. Plus and minus tolerances for location are used to some extent in industry. Therefore, they are shown here to provide some general guidance. A much better and well-defined method is to use basic dimensions to define locations and apply geometric tolerances to define the allowable variation. Position and other tolerances are explained in later chapters and should be used to replace plus and minus location tolerances.

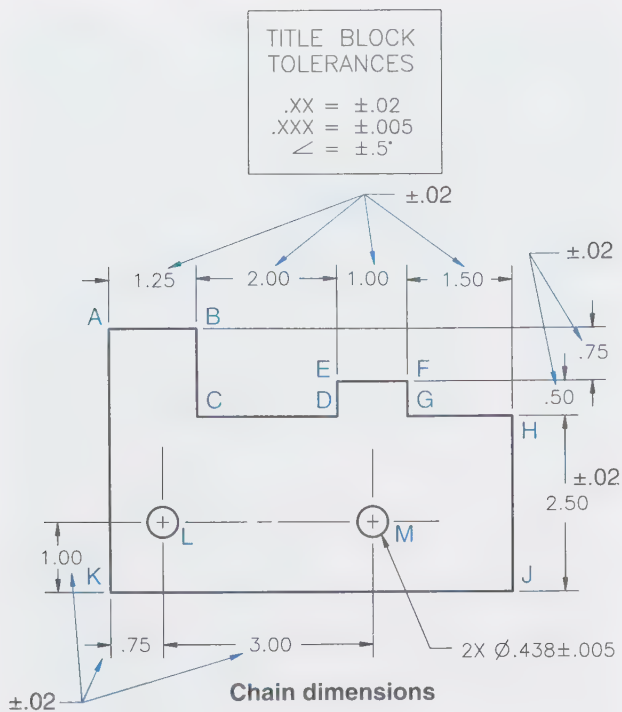
Chain Dimensioning

Chain dimensioning (point-to-point) is a system in which all features are located from adjacent features. See **Figure 3-14**. It is the most direct method for applying dimensions, but it has a definite disadvantage related to the accumulation of tolerances when using plus and minus tolerances. The location of every feature depends on the location of the feature from which it is dimensioned. If one feature is out of position by any allowable amount or has orientation variation, the location of the next feature is affected.

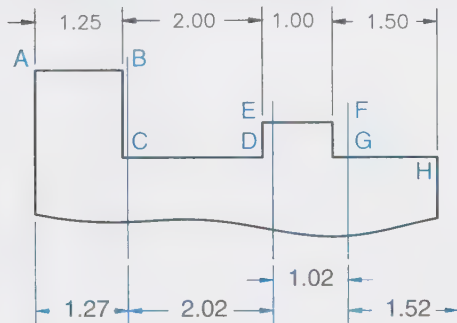
The part shown in **Figure 3-14** is dimensioned using chain dimensions. This type of part would not normally be dimensioned in this manner, but the example emphasizes the effect of *tolerance accumulation*. As shown in the figure, the title block tolerances are applicable to each dimension in the chain. The shown tolerance effects assume no presence of form or orientation variation.

Rectangular Coordinate Dimensioning

Three mutually perpendicular planes establish a rectangular coordinate system. In the *rectangular coordinate dimensioning system*, all dimensions originate from the mutually perpendicular planes. Three standard methods of showing rectangular coordinate dimensions exist. They



With all features at the nominal size, the part is 5.75 inches long.



With all features at the maximum size, assuming all features are flat and parallel, the part is 5.83 inches long.

Effect of tolerance accumulation

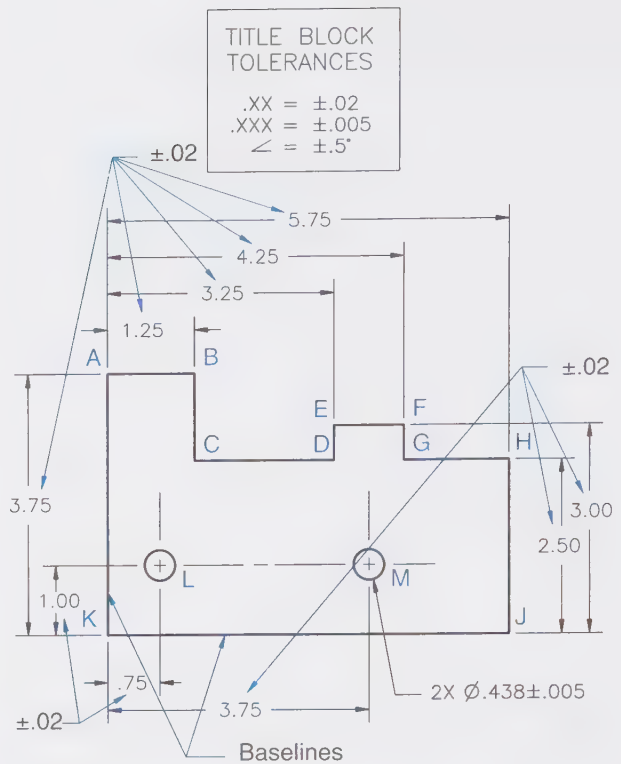
Goodheart-Willcox Publisher

Figure 3-14. Chain dimensioning results in a large tolerance accumulation when it is applied to multiple features.

are baseline dimensioning, rectangular coordinate dimensioning without dimension lines, and tabulated dimensioning.

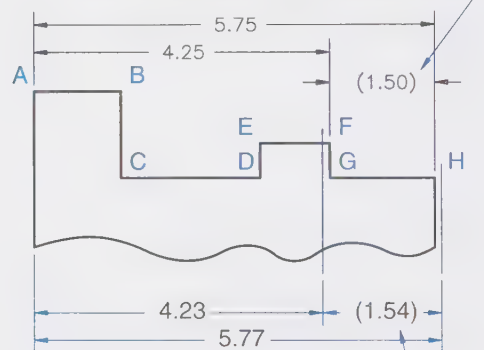
Rectangular Coordinate Baseline Dimensioning

Where location tolerances are directly applied to dimensions, *baseline dimensioning* reduces the tolerance accumulation that is caused by chain dimensioning. The accumulation is reduced because all dimensions extend from baselines. The example in **Figure 3-15** shows the application of



Baseline dimensions

With both dimensions at the nominal value, this feature is 1.50 inches wide



With both dimensions varying in opposite directions, and assuming perfect form and orientation of the features, this feature may be 1.54 inches wide (or as small as 1.46)

Effect of tolerance accumulation

Goodheart-Willcox Publisher

Figure 3-15. Baseline dimensioning can reduce the amount of tolerance accumulation on a drawing.

baseline dimensions. The shown tolerance effects assume no presence of form or orientation variation.

Two features on the part are used to establish the baselines. The left surface establishes the baseline for X coordinates (width) and the bottom

surface establishes the baseline for Y coordinates (height). All dimensions extend from these surfaces.

The title block tolerances apply to all dimensions in the given figure because no specific tolerances are used with the dimensions. All shown dimension values are two-place decimals, so their values can vary $\pm .02''$ from their respective baselines. Because all features vary in location to the baselines by $\pm .02''$, their locations relative to each other can vary by $\pm .04''$.

Rectangular Coordinate Dimensioning without Dimension Lines

In this system, dimensions are shown on a drawing without using any dimension lines. The dimension values are shown adjacent to the extension lines. See **Figure 3-16**. The dimension values shown in the figure are rectangular coordinates referenced to the origin. The origin for the coordinate dimensions is identified by zeroes placed on extension lines from the origin location. Features

located to the left of the origin have a $-X$ value, and those to the right of the origin have a $+X$ value. Features located below the origin have a $-Y$ value, and those above the origin have a $+Y$ value. Although not required by the national standards, it is common practice to draw arrows from the 0,0 extension lines to show the $+X$ and $+Y$ directions.

When using aligned dimension values in this system, all dimension values are shown so that they appear above the extension lines relative to the viewing direction. When using aligned dimensions, all dimension values read from the bottom or right side of the page. Sufficient space must be provided between the extension lines to allow the dimension value to be entered and clearly associated with only one extension line. A *joggle* (offset) may be required in some extension lines to provide sufficient space for the dimension value.

The dimension values may also be placed at the end of the extension lines. This reduces the required space between extension lines.

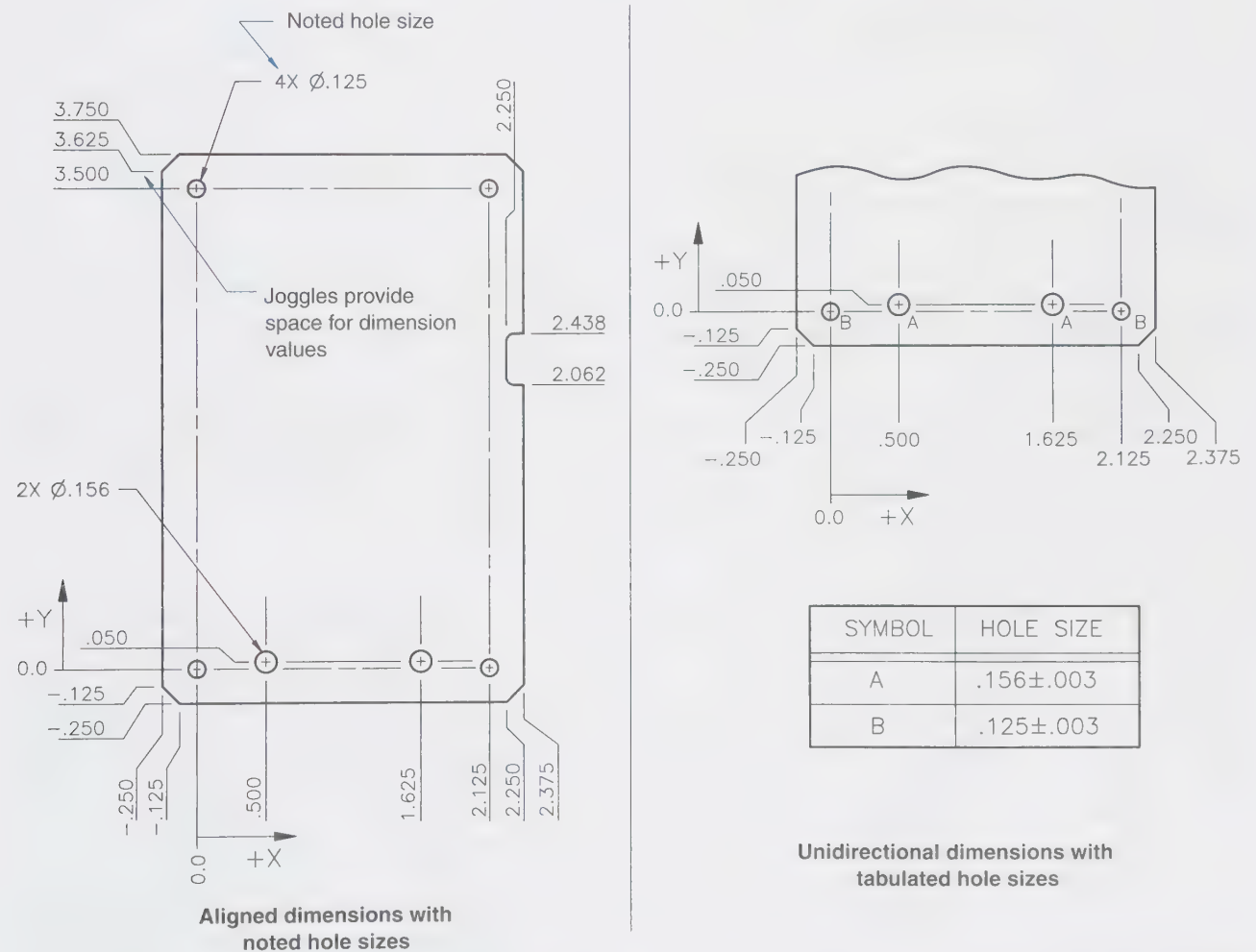


Figure 3-16. Rectangular coordinate dimensioning without dimension lines shows the dimension values near the ends of the extension lines.

Unidirectional dimension values require more space between extension lines to prevent the dimension values from running together. More joggles must be inserted into the extension lines to provide adequate clearance, and many of the joggles will be offset by a greater distance. The increased amount of offset in the joggles can make unidirectional dimensions hard to read when closely spaced features exist on the part.

Hole sizes may be noted on the part or referenced to a hole size table. When a table is used, the hole on the part is labeled with a letter that corresponds to a size shown in the table. For example, two holes labeled A in [Figure 3-16](#) are .156" in diameter and the diameter tolerance is $\pm .003$ ".

Rectangular Coordinate Dimensioning Using Tabulated Dimensions

Locations and sizes for a large number of holes can be conveniently shown in a table. See [Figure 3-17](#). A table showing hole locations and sizes provides a well-organized and precise system for dimensioning a large number of holes. All hole location values are related to a common origin.

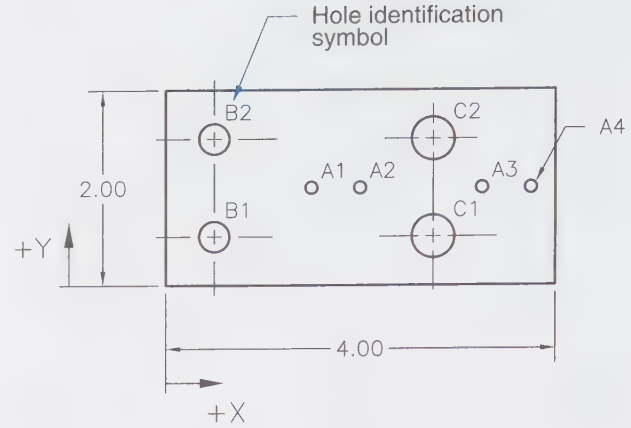
Tables used to show hole locations and diameters are not standardized. Any arrangement that clearly shows all data is acceptable. The table used as an example in [Figure 3-17](#) is one acceptable format. For more complex parts, hole positions may require X, Y, and Z coordinates to specify location and vector information to specify the angle of the holes. A table is typically not used if hole size and location are defined by CAD model data rather than by a drawing.

A **hole identification symbol** is shown in one column to identify a hole on the part. The locating X and Y coordinates are shown in the column adjacent to the symbol. The hole size is listed in the next column, and the size tolerance for the hole diameter is shown in the last column.

The hole identification symbol in the example includes a letter and number. The letter represents all holes of a particular size. The number identifies a specific hole of the size represented by the letter. It is not required that this system of hole identification be used.

A geometric symbol may be used to represent a hole size in place of the letter and number shown in the figure. As an example, a circle with a horizontal line across it might represent each .093" hole, while hexagons might represent the .125" diameter holes.

Tabulated dimensions may be used to dimension the outline of an object. If a table is used,



DRILL TABLE

SYMBOL	LOCATION		SIZE	TOL
	+X	+Y		
A1	1.50	1.00	.125	+.005 -.000
A2	2.00	1.00		
A3	3.25	1.00		
A4	3.75	1.00		
B1	.50	.50	.312	+.005 -.000
B2	.50	1.50		
C1	2.75	.50	.438	+.006 -.000
C2	2.75	1.50		

Goodheart-Willcox Publisher

Figure 3-17. Hole locations and sizes may be shown in a table.

features along the outline are identified with numbers, letters, or symbols. See [Figure 3-18](#). The identified points are called **station points**. On curved outlines, station points are located close enough together to obtain the required accuracy for the curve. The station points are shown in a table along with the located coordinates for each point. A curved surface may also be dimensioned by specifying the mathematical formula that defines the curve.

Polar Coordinate Dimensioning

The **polar coordinate dimensioning system** requires that the location definition include an angular dimension. For some parts, angular location of related features is a better method than rectangular coordinate location. Angular relationships are often important for rotating parts.

The part shown in [Figure 3-19](#) includes several features that are dimensioned through specification of polar coordinates. The two holes are dimensioned by an angle showing the distance from the edge of the part to one hole. Another angle is specified for the distance between the two holes. The location of the two holes is further defined by a

radius dimension. Omission of the radius dimension or either of the two angle dimensions would result in incomplete definition of the hole locations.

Direct (Combined) Dimensioning Systems

When using plus and minus tolerancing for location, the combined use of baseline dimensioning and chain dimensioning systems allows the tolerance accumulation on a drawing to be minimized. Another option is to use datums and profile

tolerances to achieve the same effect without any of the ambiguity that is sometimes created where tolerances are directly applied to the location dimension values.

The sheet metal bracket in Figure 3-20 shows the combination of the two systems. The bracket is designed to position a machine block at a specific height and orientation. The block height is controlled by the height of the attachment screw, and its orientation is controlled by an alignment pin. The bracket has two mounting holes in the bottom flange.

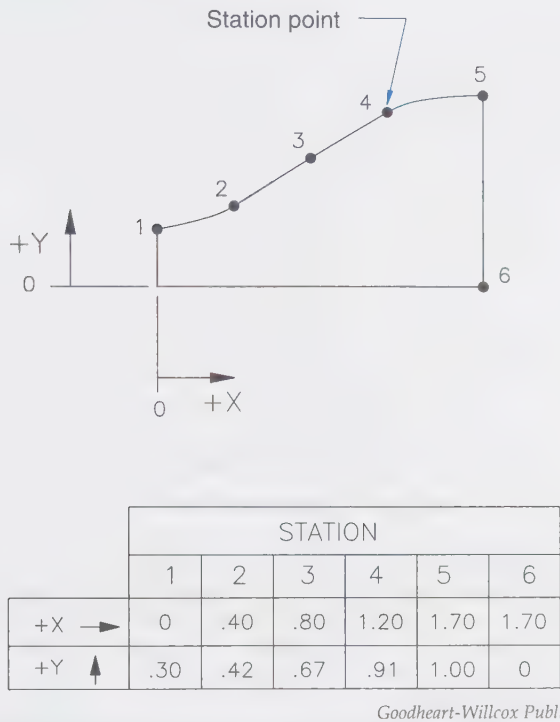


Figure 3-18. Station points may be used when outline dimensions are shown in a table. Dimensions in a table are known as tabulated dimensions.

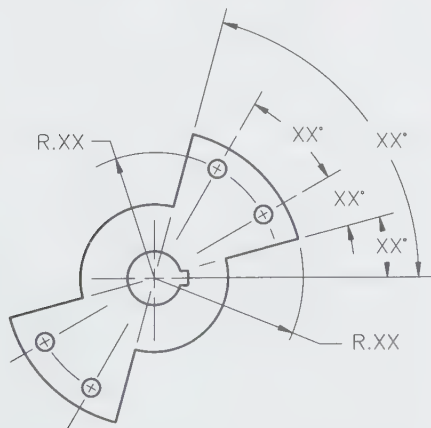


Figure 3-19. Polar coordinate dimensions combine angular and distance dimensions to define sizes and locations.

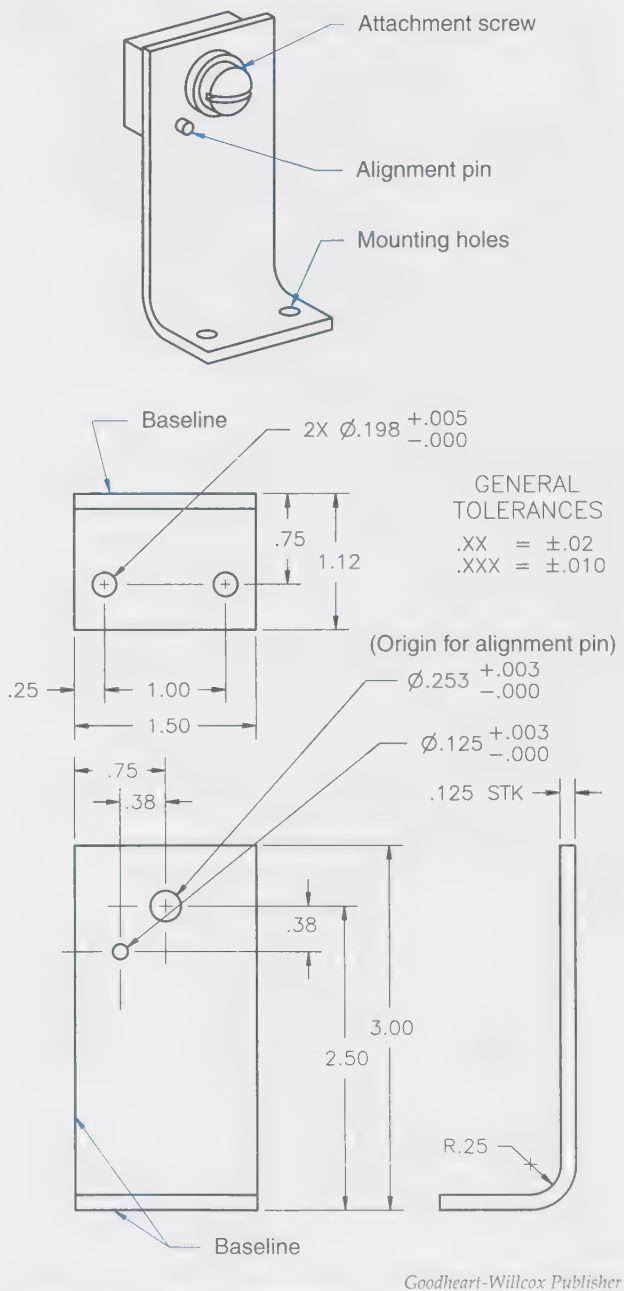


Figure 3-20. A combination of baseline dimensioning and chain dimensioning can reduce tolerance accumulation on a part.

The mounting surface of the bracket is used as a baseline from which the attachment screw hole is located. This determines the height of the block relative to the mounting surface. The same baseline is used to show the overall height of the bracket. The left side of the bracket is used as a baseline from which the attachment screw hole is located in the width dimension. The same side of the bracket is also used as a baseline for locating one of the mounting holes and to dimension the width of the bracket.

The back of the bracket is used as a baseline from which the mounting holes are located and to show the overall depth of the bracket. The back surface of the bracket is selected as a baseline to ensure a controlled location of the machine block relative to the mounting holes. The back surface location relative to the mounting holes can only vary by $\pm .02$ " from the .75" specified.

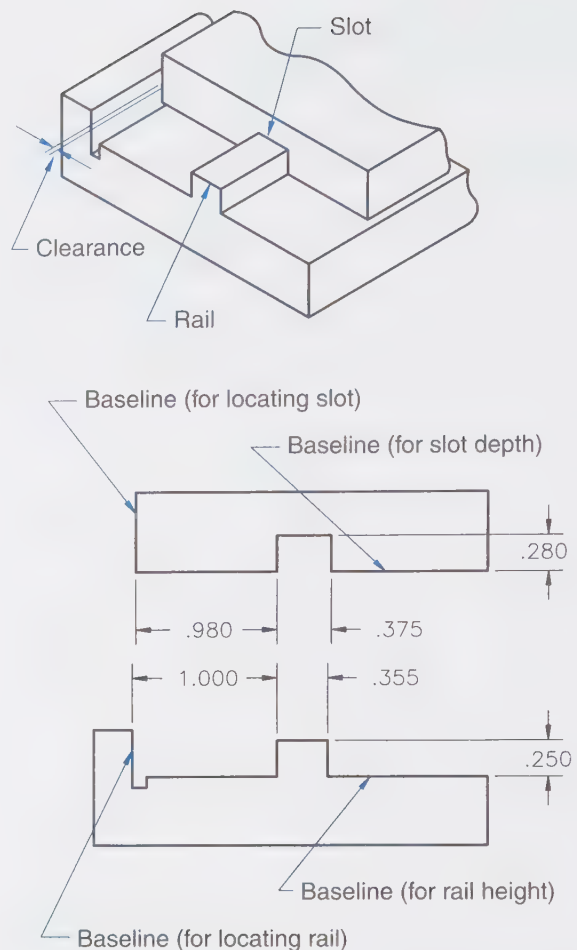
The alignment pin hole is located from the attachment screw hole. This ensures a minimum tolerance between the two holes. Minimizing tolerance helps ensure that the alignment pin can go through both the bracket and machine block while the block is attached to the bracket.

One hole within a hole pattern is often located and then each hole in the pattern is located from the first one. This is in effect using the first hole as an origin for the other holes. The attachment screw hole and alignment pin hole in the given bracket can be thought of as a hole pattern containing two holes.

Parts should be dimensioned using functional features as baselines. See **Figure 3-21**. On the given part, the 1.000" location dimension for the rail is from a surface that must be cleared by the part that slides on the rail. The use of this surface as a baseline reduces the tolerance accumulation between it and the rail. Clearance between the baseline surface and the sliding part is only affected by one dimension on each of the shown parts.

The rail width is dimensioned from the located side of the rail, rather than by a second dimension from the rail locating baseline. This reduces the tolerance on the rail width to the variation applicable to the one dimension. Reducing the tolerance on the rail helps to ensure that it will fit into the mating slot.

Locating features from functional baselines reduces the tolerance accumulation on their locations. Using chain dimensions to define feature size reduces tolerance accumulation on the individual feature.



Goodheart-Willcox Publisher

Figure 3-21. Functional baseline features are often used where dimensioning the locations of related features.

Pro Tip

Avoiding Plus and Minus Tolerances on Location Dimensions

The application of plus and minus tolerances on location dimensions as explained above has some risk of poorly defined part requirements and should be avoided on new drawings. The primary problem with plus and minus tolerances on location dimensions is caused by form or orientation variation that will exist on parts. When form or orientation variation exists on a part, questions without answers can arise regarding the correct origin and measurement direction. These practices are being replaced by the use of basic dimensions and geometric tolerances as explained in the following chapters.

Dimension Placement in Multiview Drawings

General guidelines must be followed when applying dimensions on multiview drawings. These guidelines ensure that size and location information is presented in a clear and consistent manner. Following dimension placement guidelines further clarifies a drawing and makes it easier to read.

Regardless of the dimensioning system used, such as chain or rectangular coordinate systems, the following general guidelines are applicable.

- Dimension where the feature contour is shown.
- Dimension between the views.
- Dimension off the views (outside the object).
- Dimension with consideration given to how the parts are assembled.
- Consider the fabrication processes and capabilities.
- Consider the inspection processes and capabilities.
- Create a logical arrangement of dimensions.
- Stagger dimension values.
- Avoid dimensioning to hidden lines.

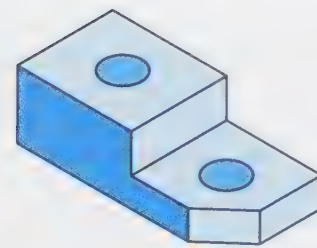
All dimensioning guidelines are based on common sense and industrial processes. As knowledge in machine shop and inspection practices is gained, the dimensioning guidelines become more natural and require a minimal amount of effort to remember.

Correct dimension application requires that the location and size for every feature be shown. This means that the height, width, and depth must be given for each feature and that the location of the feature on the part must also be given. To ensure that a part is completely dimensioned, the part can be mentally broken down into the geometric shapes that form the part.

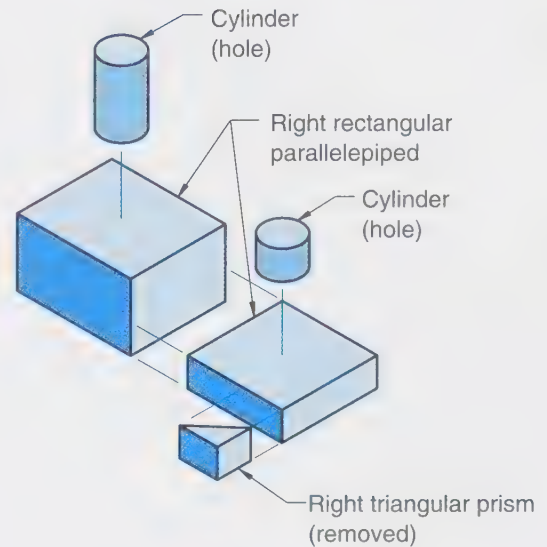
A step block is shown in [Figure 3-22](#). The step block consists of five geometric shapes. Two rectangular blocks create the basic shape of the part. Two cylinders and a right triangular prism are removed from the part to complete the step block. The height, width, and depth dimensions for the two blocks are given to outline the object. The locations and diameters for the two holes are specified. The size of the right triangular prism is specified by locating two corners. The third corner of the triangle is the removed corner of the step block.

Dimensioning Feature Contour

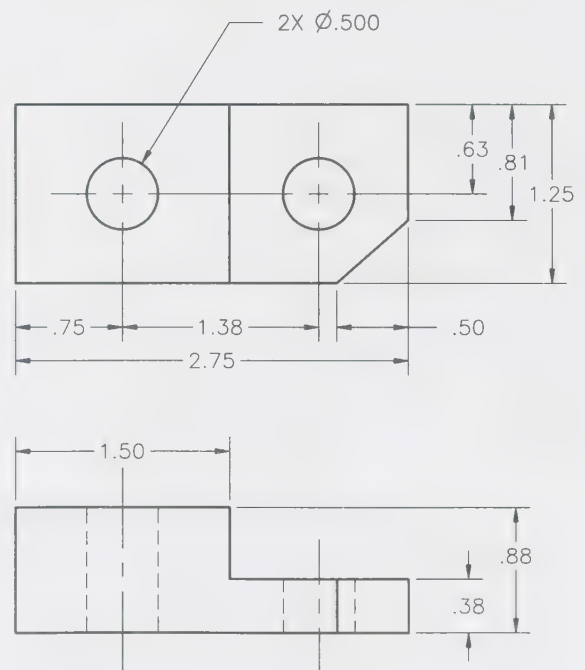
Dimensions are applied where the contour of a feature is best shown. This permits the dimension



Step block



Geometric composition



Required dimensions

Goodheart-Willcox Publisher

Figure 3-22. Mentally breaking a part into the geometric shapes that form the part is one method for determining the dimensions needed on a drawing.

and the feature contour to be viewed simultaneously. When the dimensions are not applied to the contour, the dimensional requirements are not as clear. See **Figure 3-23**. The upper half of the figure shows the dimensions correctly applied and the lower half shows the same part with poorly applied dimensions.

The front view of the given object shows a distinct offset in the height of the part. The offset appears as a line across the top view and does not show any distinct change. The offset is dimensioned in the front view as 1.00" from the left side of the object. The .38" height of the offset is also dimensioned in the front view.

An angled surface on the part removes one of the square corners. The location of the angled surface is dimensioned in the top view because the change in contour is clearly seen there, and the corner appears only as a line between two surfaces in the front view.

Dimensions applied to the wrong views force the reader to look away from the dimension to see

a view where the contour is shown. This is time consuming and can result in errors. Placement of each dimension on the contour of the feature allows the size or location to be determined without as much chance for error.

Dimensioning between Views

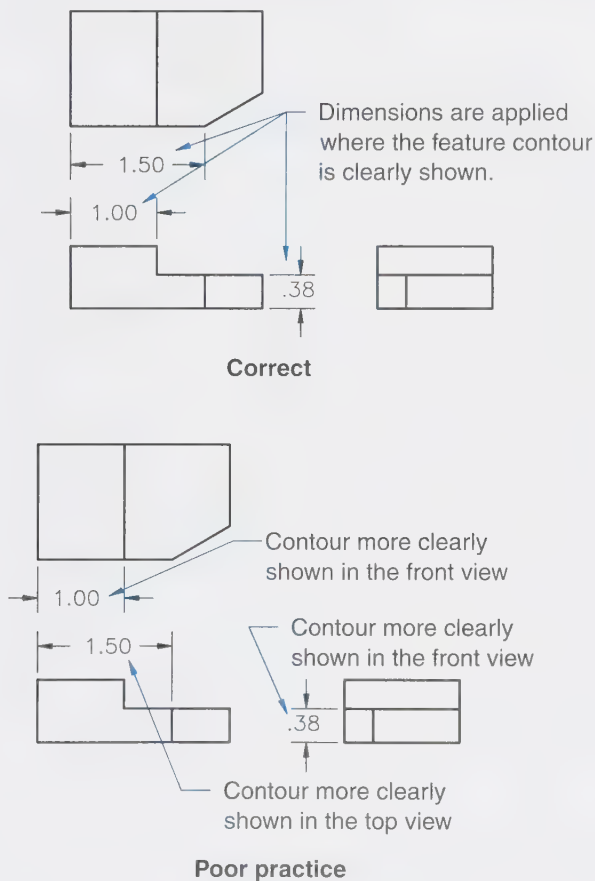
Dimensions of size and location should be placed between views when practical. This general guideline is followed for all dimensions that can be applied between the views without causing a loss of drawing clarity. A dimension should not be placed between views if it requires running extension lines across several features within a part. See **Figure 3-24**. The left half of the figure shows dimensions in the preferred location between the views. The right half shows the same dimensions applied where they are not between the views.

The extension lines used to show the dimensions between the views do not cause any loss in drawing clarity when correctly done. Dimensions applied between two views allow the dimensions to be easily related to both views. Extension lines for any one dimension generally extend to only one view, but the placement of the dimension allows it to be related to both views. If dimensions are not placed between the views, the view on which the dimension is applied separates the dimension from the second view.

Dimensioning off the Views

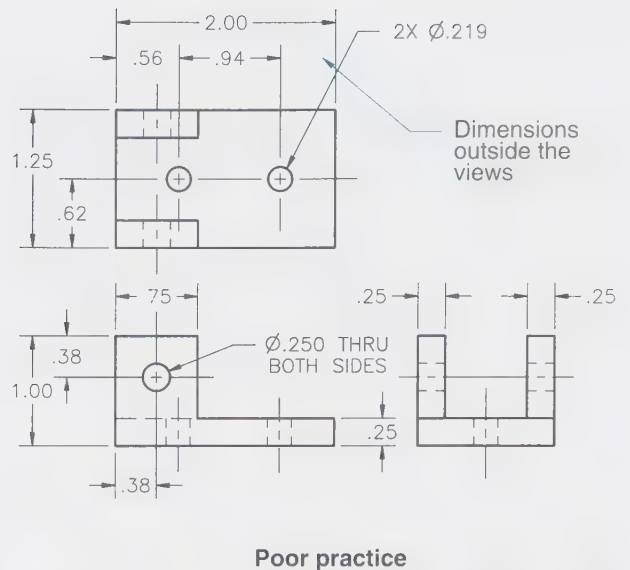
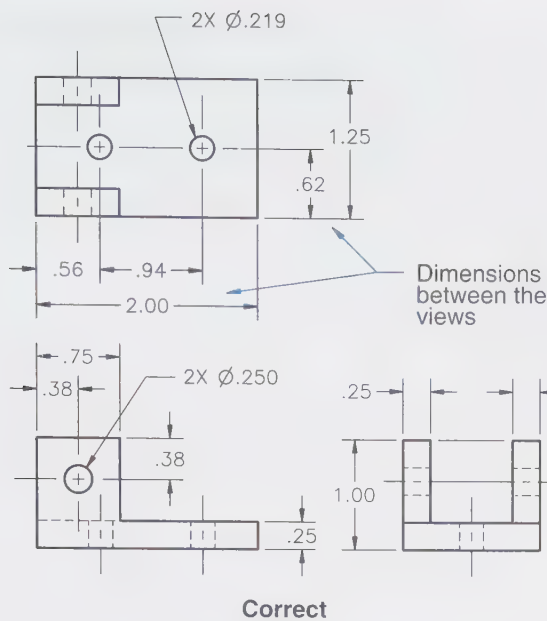
Dimension lines are not to terminate on object lines if it is possible to clearly dimension the feature using extension lines. Extension lines extend the feature outside the object to allow placement of the dimension lines on the outside of the object. See **Figure 3-25**. The example in the upper half of the figure shows the correct way in which extension lines are used to locate dimensions off the views. The lower half of the figure shows that incorrect use of extension lines can result in dimensions that are inside the object and dimension lines that terminate on object lines.

There are exceptions that permit placement of dimensions within a view. For a large part or assembly where long extension lines would negatively impact clarity, a dimension may be placed within the outline. If the extension lines would cross many other features or possibly coincide with multiple features, it may be appropriate to use a dimension within an outline. Exceptions must be addressed in a manner that does not diminish clarity.



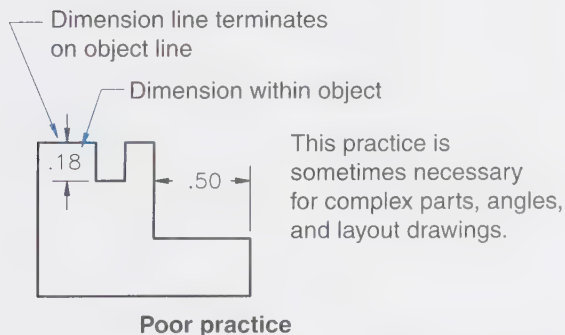
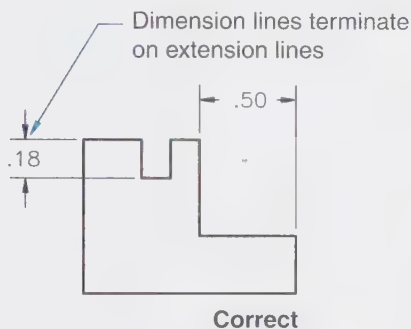
Goodheart-Willcox Publisher

Figure 3-23. Features are most clearly dimensioned in the view that shows the feature contour.



Goodheart-Willcox Publisher

Figure 3-24. Dimensions are placed between views when possible.



Goodheart-Willcox Publisher

Figure 3-25. Dimensions are normally placed off the object. Avoid placing dimensions on the object.

Dimensioning According to Related Parts

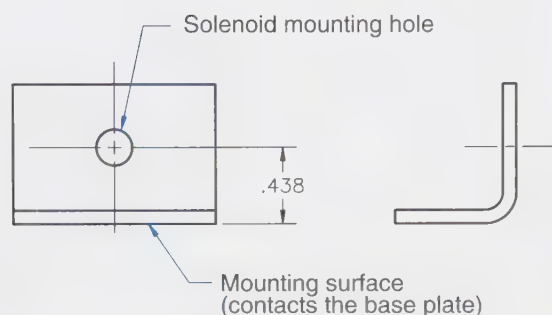
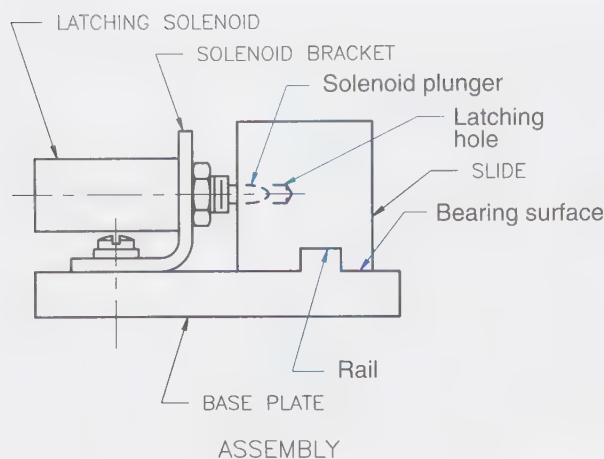
Parts that must fit into an assembly are dimensioned with consideration of function and the way in which the parts fit together. See Figure 3-26. Four parts from a simple assembly are shown. They are a base plate, slide, latching solenoid, and

solenoid bracket. The slide moves on the base plate and is guided by a rail. The latching solenoid is mounted on the solenoid bracket, and the solenoid bracket mounts on the base plate.

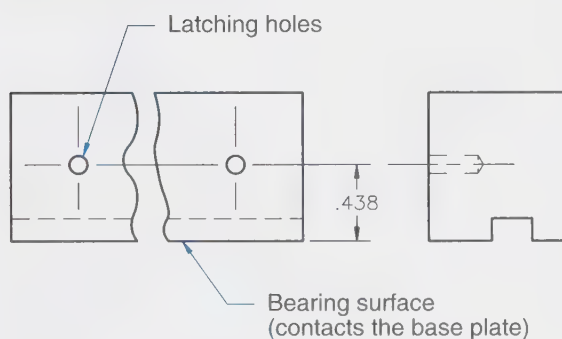
The solenoid bracket must locate the solenoid in a position that will allow the solenoid plunger to enter the latching holes in the slide. This will occur only if the latching holes are located the same distance from the base plate as the solenoid plunger. To accomplish this, the solenoid mounting hole is dimensioned from the bottom surface of the solenoid bracket. The latching holes are dimensioned from the bearing surface (bottom) of the slide. The surfaces used as baselines on the two parts are both in contact with the base plate when the parts are assembled. Locations for the holes on two different parts therefore originate from a common baseline in the assembly. Use of a common baseline for two parts ensures a minimum amount of tolerance accumulation between the two parts.

General Arrangement of Dimensions in Multiview Drawings

Dimensions are arranged to minimize crossing of extension and dimension lines. See Figure 3-27. Placing large dimensions farther from the object than small dimensions is one way to minimize crossing lines. In the given example, the "Correct" application shows the effect of placing large dimensions outside the smaller ones. All size and location dimensions for the stepped groove are clearly shown.



SOLENOID BRACKET

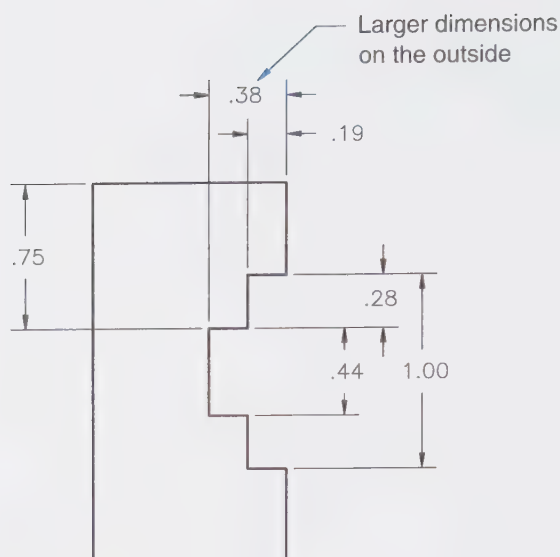


SLIDE

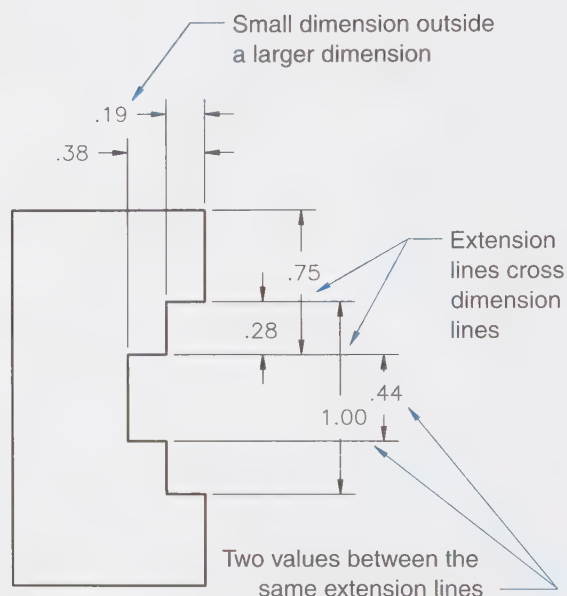
Goodheart-Willcox Publisher

Figure 3-26. The location of the solenoid mounting hole is dimensioned from the same baseline as the latching holes. Related features on separate parts should be dimensioned from a common baseline.

When the same stepped groove is dimensioned with small dimensions outside larger ones, the size and location information isn't as clear. Dimension and extension lines cross because of the poor dimension arrangement. It is difficult to determine the point of application for a dimension if the dimension line crosses several extension lines. Placement of two or more values between



Correct



Poor practice

Goodheart-Willcox Publisher

Figure 3-27. Dimensions are arranged to avoid crossing dimension and extension lines. This normally requires that the large dimensions be placed outside smaller ones.

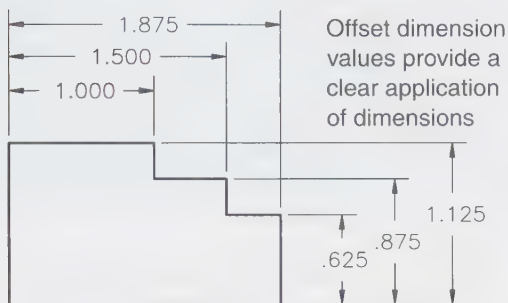
the same extension lines can become necessary when small dimensions are placed outside larger ones, possibly causing confusion as to which dimension value is applicable. Confusing dimensions are avoided by using a general arrangement where small dimensions are placed closer to the object than large dimensions.

Staggered Dimension Values

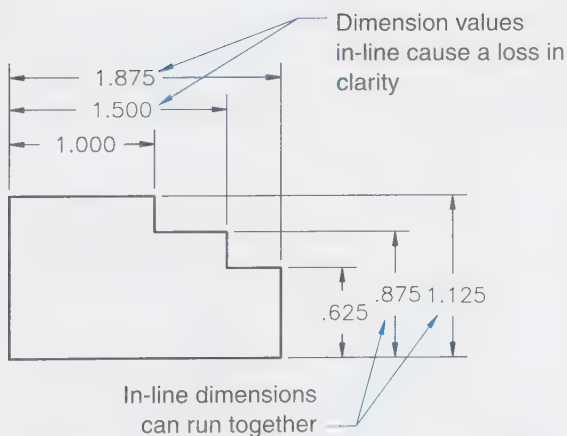
Staggering dimension value locations helps to more clearly show the values when several rows of dimensions are used. See **Figure 3-28**. There is a general requirement, as stated earlier, that the dimension value be centered, but ensuring readability is of greater importance. Each value is placed near the center of its dimension line, but enough offset is used to avoid putting the values directly in-line with one another. Failure to stagger vertical dimensions can result in dimension values that run together. The dimension spacing may need to be increased if software limitations prevent movement of the dimension value.

Dimensioning Hidden Features

Generally, it is not acceptable to dimension a feature where it is shown using hidden lines. It is preferable to dimension the feature where it is drawn as an object line. See **Figure 3-29**. If the feature appears hidden in all views, a section view is required. The section can be a broken-out section



Correct



Poor practice

Goodheart-Willcox Publisher

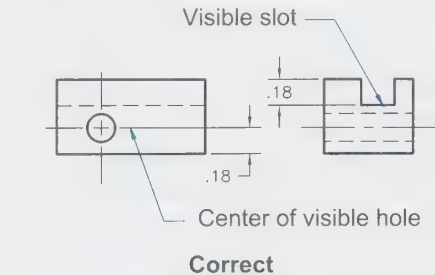
Figure 3-28. Staggered positions for dimension values make it easier to read the dimensions.

that shows the feature with object lines, or it can be one of the other types of section views.

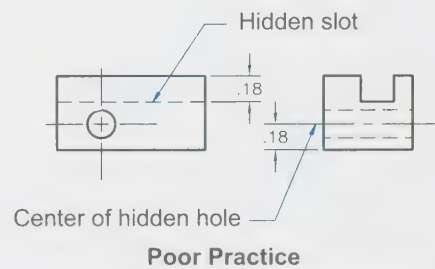
Dimensions in a half section view can extend from an object line to a hidden line when showing the size of a feature. See **Figure 3-30**. This is acceptable because one side of the view shows the feature as visible, and the other side shows the feature as hidden.

Double Dimensioning

It is generally not a good practice for the size or location of a feature to be controlled by two sets



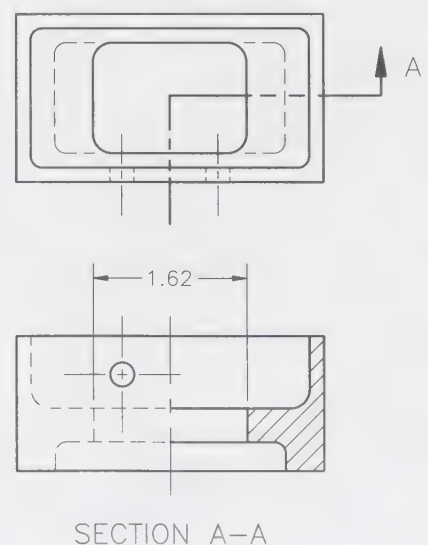
Correct



Poor Practice

Goodheart-Willcox Publisher

Figure 3-29. Features are dimensioned where they are shown as visible.



SECTION A-A

Goodheart-Willcox Publisher

Figure 3-30. Dimensions may extend to the hidden portion of a feature shown by a half section.

of dimensions that may result in two conflicting requirements. In **Figure 3-31**, a “Correct” application and a “Poor practice” are shown. In the “Correct” application, the size of each feature can be determined in only one way.

Distances A,B; B,E; and A,F have dimensions applied directly to them in the illustration labeled as “Correct.” Title block tolerances specify the acceptable variation for each dimension. Distance E,F is not directly specified in the “Correct” example; it does not require a dimension. When all dimensioned features are produced to the sizes specified, distance E,F will be made to a definite size. The permissible range of sizes at which distance E,F may be produced is determined by the tolerances on the other features (which are dimensioned) and can be calculated through the shown procedure.

The accumulation of tolerance on distance E,F can be controlled by dimensioning the part

differently if the dimensions in the given figure are unacceptable. However, no more than three dimensions may be used. Adding a fourth dimension creates a double-dimensioned feature.

The illustration labeled “Poor practice” shows the features double dimensioned. **Double dimensioning** results in a condition where the size or tolerance for a feature can be determined in more than one way.

Four dimensions are used to define distances A,B; B,E; E,F; and A,F in the “Poor practice” example. This is one dimension too many. A title block tolerance of $\pm.02$ ” is applied to each dimension. This tolerance applied to each individual feature conflicts with the tolerance accumulation that occurs when all other features are considered.

Examination of distance A,F shows how a conflict in tolerances occurs when double dimensions are used. A dimension of 2.25” with a tolerance of $\pm.02$ ” is directly applied to distance A,F. This

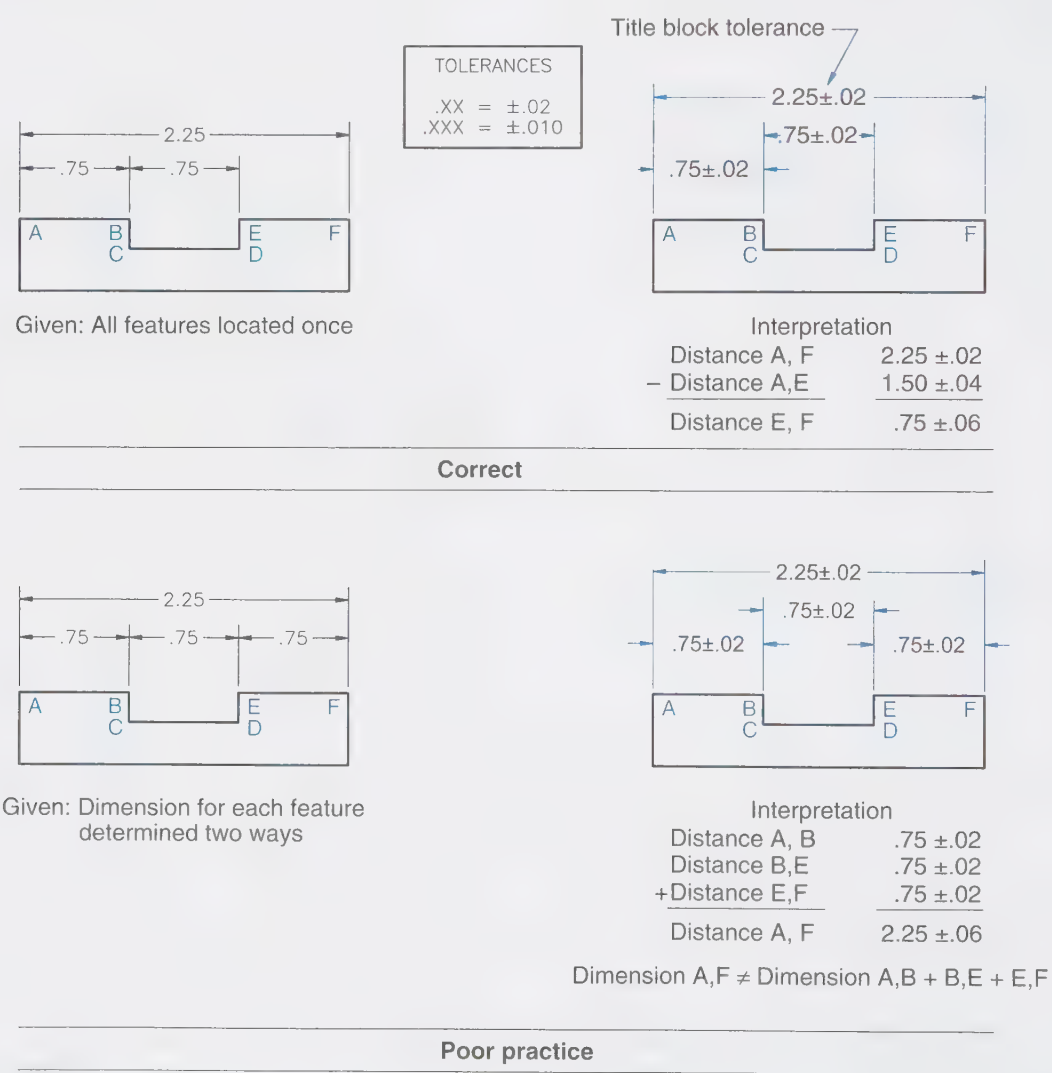


Figure 3-31. Double dimensioning causes an unacceptable conflict between the tolerance accumulations on a part.

provides one requirement for distance A,F. A second requirement for distance A,F is determined by adding the dimensions applied to distances A,B; B,E; and E,F. The sum of these dimensions is 2.25" with a total tolerance accumulation of $\pm .06$ " for distance A,F.

The directly applied dimension for distance A,F requires a tolerance of $\pm .02$ " and the derived requirement from the other features is $\pm .06$ ". Because these values are not equal, there is a conflict in the two requirements. It is generally not acceptable to place two conflicting requirements on a feature. There may be exceptions as explained below.

Pro Tip

Double Dimensioning Exception

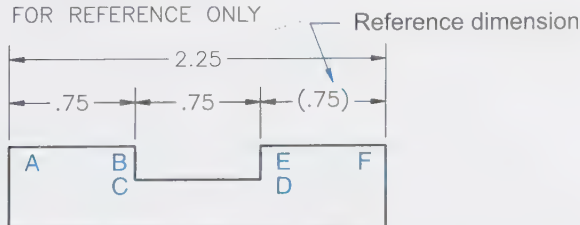
Should a situation exist where the overall tolerance must be specified and each individual tolerance is also important, a note may be applied that says all tolerances must be achieved. This has the impact of requiring the manufacturer to control variation in a way that prevents tolerances from accumulating. A note such as the following is recommended.

FABRICATION TOLERANCES ON INDIVIDUAL FEATURES MUST NOT ACCUMULATE IN A MANNER THAT VIOLATES THE OVERALL TOLERANCE.

Reference Dimensions

It is possible to apply a *reference dimension* to a feature that is already controlled by other dimensions. See Figure 3-32. Reference dimensions are used only for what the name implies; they are used as a reference that shows the as-designed size or location, but they do not indicate the accuracy of the dimension. Because no tolerance is implied by a reference dimension, its use does not indicate

NOTE:
INFORMATION SHOWN
IN PARENTHESES IS
FOR REFERENCE ONLY



Goodheart-Willcox Publisher

Figure 3-32. A reference dimension shows an as-designed size or location. No tolerance is applicable to a reference dimension.

double dimensioning. Tolerance for the affected feature is determined from the other dimensions on the drawing.

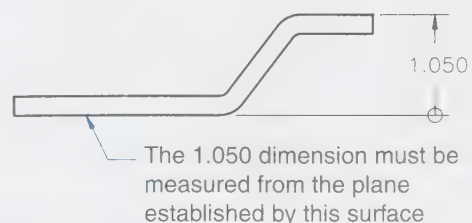
Figure 3-32 shows the same object that was shown in Figure 3-31. Distance E,F in Figure 3-32 is controlled by the dimensions on distances A,F; A,B; and B,E. The as-designed size of distance E,F is shown as a reference dimension. The dimension is only provided to show the approximate size of distance E,F. It is not meant to be a number that must be used while making the part. If a person does use a reference dimension when making a part, the other dimensions must be produced within the acceptable variations allowed by specified tolerances.

Indicating a Dimension Origin

Identification of the *dimension origin* is sometimes required when variations on a part could cause a difference in how features are measured for acceptance or rejection of produced parts. See Figure 3-33. The given example shows a height dimension between offset surfaces. One of these surfaces is relatively long and the other one is short. If each of these features includes angular errors relative to the vertical leg of the part, selection of which surface acts as an origin can impact whether or not parts are acceptable. A small angular error on the short horizontal leg can have a significant impact on the measured height of the part, if the short leg is used as the origin.

To make certain the correct feature is used as the origin for the height dimension, the origin can be identified. This is done by replacing one of the dimension arrowheads with a circle. The circle is placed at the end of the dimension line that is to act as the origin.

Identification of an origin by this means should only be used when measurement relative to a specific origin is required, and then only if the part is not controlled by geometric tolerances referenced to a datum reference frame. Datum reference frames are described in the chapter on datums and datum feature references.



Goodheart-Willcox Publisher

Figure 3-33. A dimension origin may need to be indicated to ensure the desired design requirement is met.

Dimensions Applied to Special Views

The methods and guidelines for dimensions are applicable to all views in a multiview drawing, including auxiliary and section views. When CAD models are used to generate the orthographic views, pictorial drawings are sometimes included in a multiview drawing. The pictorial views may include dimensions.

Dimensioning an Auxiliary View

Auxiliary views are used to show the true shapes of features on inclined and oblique surfaces. It is required that dimensions be applied where the feature is seen in true size. Features

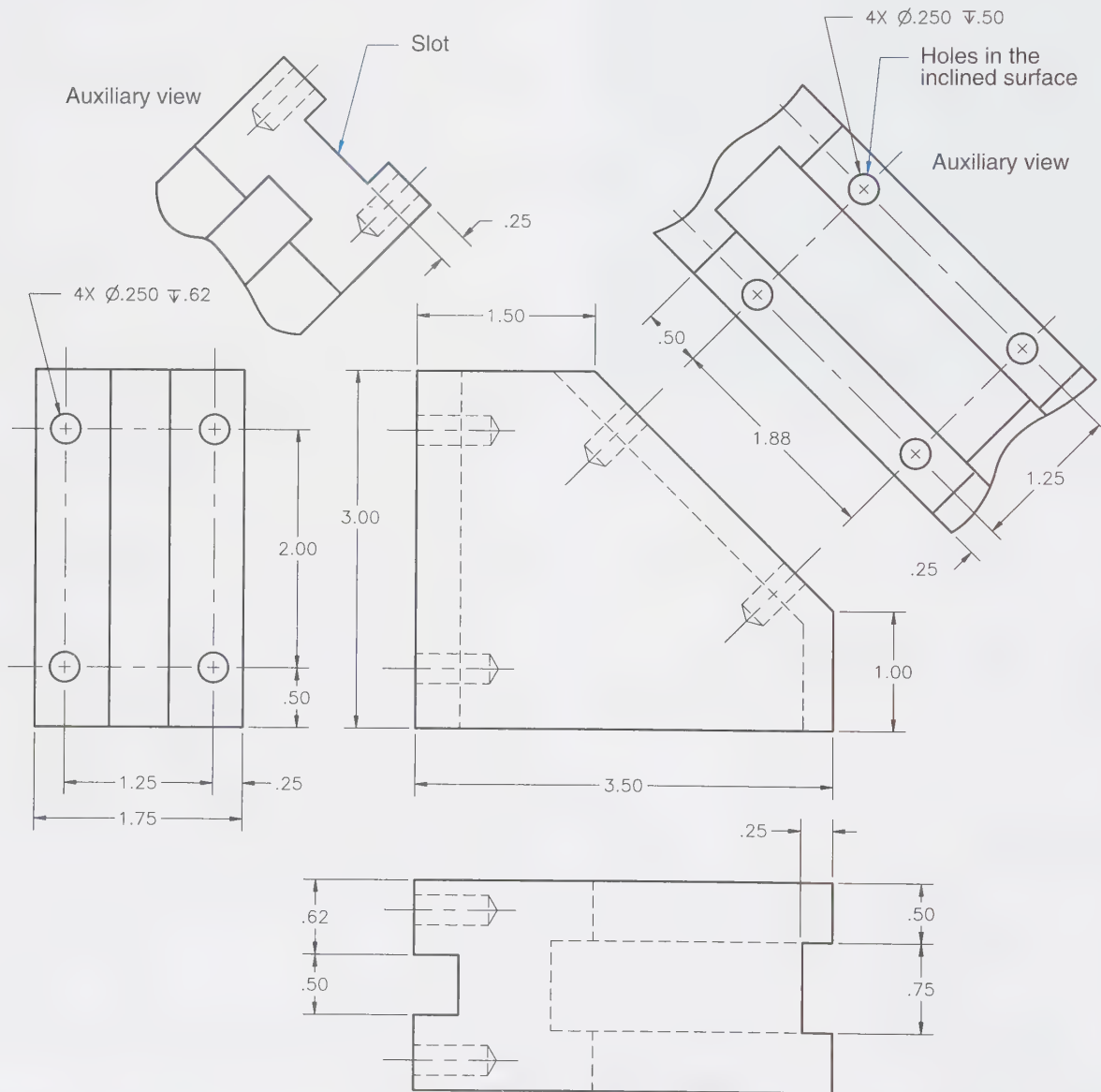
shown true size only by an auxiliary view must be dimensioned in the auxiliary view. See [Figure 3-34](#). Features shown true size by principal views should be dimensioned on a principal view.

The given example has an inclined surface with four drilled holes. A slot is cut into the inclined surface. True shape and size for the inclined surface and slot is shown in auxiliary views.

Dimension lines in the auxiliary views are parallel to the distances being specified. Extension lines are perpendicular to the dimension lines.

Dimensioning a Section View

Section views are dimensioned in accordance with all dimensioning requirements previously explained for principal views and auxiliary views.



Goodheart-Willcox Publisher

Figure 3-34. Dimensions are shown on an auxiliary view only when they cannot be shown clearly on the principal views.

Section lines (crosshatch lines) sometimes create a need for special consideration when dimensioning. Extension lines and section lines are drawn with the same line type. To avoid confusion between the two lines, a break in the section lining may be made near the end of an extension line. See [Figure 3-35](#). Section lines may also be broken around arrowheads that fall within a sectioned area.

Dimensioning Pictorial Drawings

Three-dimensional CAD models and pictorial views of those models are often dimensioned to the extent needed to convey information that is not intended to be obtained directly from the CAD model. A CAD model is not considered a pictorial drawing, but the screen image or a printed copy may appear to be a pictorial view. The following does not apply to dimensioning a CAD model. The following also does not apply to a pictorial view created directly from a dimensioned CAD model.

The following only applies to pictorial views created in one of the formats defined by ASME Y14.3. (Prior to 2012, pictorial view requirements were defined by ASME Y14.4.) Those formats include axonometric, oblique, and perspective views. Each format has subcategories. Axonometric formats include isometric, dimetric, and trimetric.

Dimensions on pictorial views are clearest when all the dimensions running in one direction are

shown parallel to a single plane. See [Figure 3-36](#). All width dimensions in an oblique view may be shown on either the horizontal or frontal planes. It is best to only use one of the two planes. Depth dimensions may be shown on either the horizontal or profile planes. Height dimensions may be applied on the frontal or profile plane. All width dimensions in the given oblique view are shown parallel to the frontal plane, all height dimensions parallel to the frontal plane, and all depth dimensions parallel to the horizontal plane.

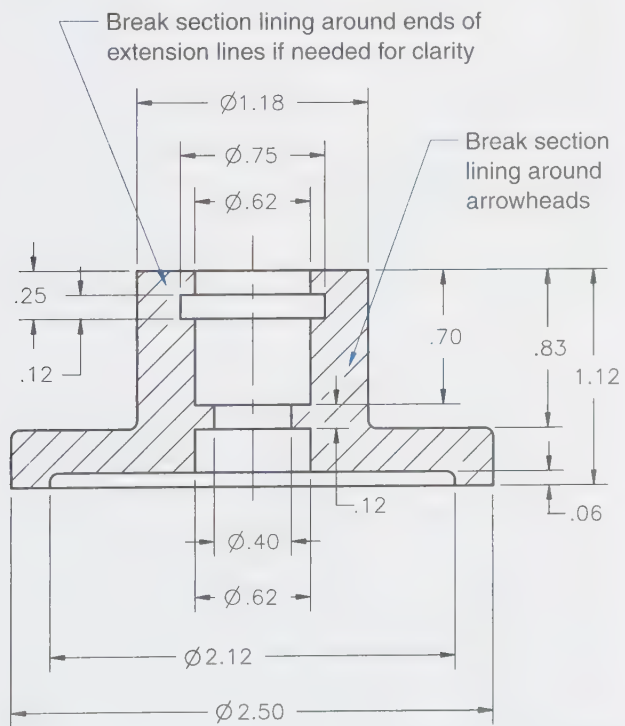


Figure 3-35. Internal features are dimensioned in section views.

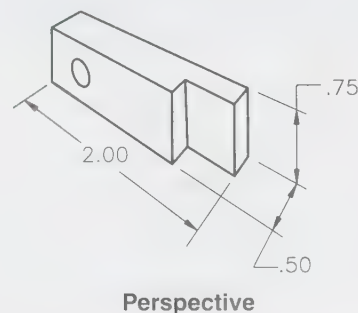
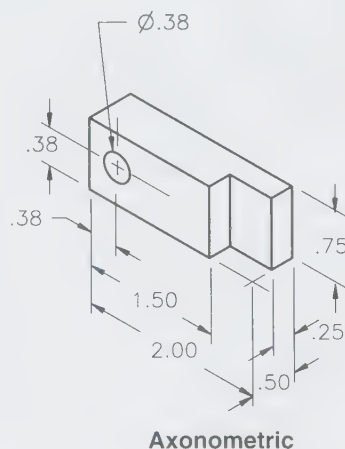
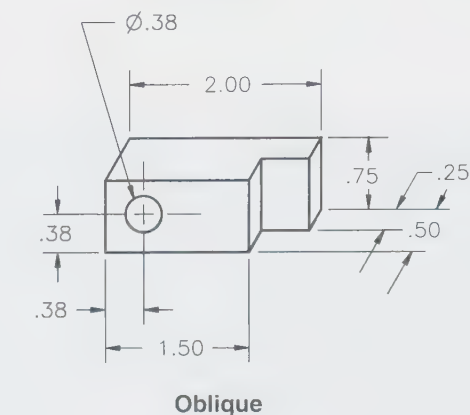


Figure 3-36. Dimensions applied in pictorial views should lie on a plane that results in clear definition of requirements.

Axonometric views are dimensioned following the same general rules as applied to oblique views. All dimensions in one direction should lie parallel to one plane.

Perspective drawings are seldom dimensioned with more than overall dimensions. The dimension lines in a perspective view recede toward the vanishing points used for the object lines.

All three examples in the given figure show the dimension values in a horizontal orientation. This practice is not required for pictorial views created from CAD models. Pictorial views that are generated from CAD models have specific dimensioning requirements defined by ASME Y14.41.

Standards Advisory

Dimensioning Pictorial Views

The use of dimensioned pictorial views is increasing. Applicability is not always clear regarding dimensioning requirements for pictorial views as shown in ASME Y14.41. Standards teams are working to update standards to improve on this. View requirements for orthographic and pictorial views are now in one standard, ASME Y14.3. Dimensioning practices are defined in ASME Y14.5 and Y14.41 with some specialty applications—such as castings, forgings, and molded parts—in ASME Y14.8.

Dimensioning with Notes

Information that cannot be clearly shown through the views, dimensions, symbology, or CAD model must be put into notes or referenced documents. Any written information that is made part of a drawing or model is considered a drawing note. Notes may be applied on the field of the drawing or put in a separate notes document.

The type of information given in a note may be general and apply to an entire drawing. It is also possible for a note to be very specific and apply only to a single feature or area on a part.

Notes are an important part of product definition whether it is a drawing or a CAD model. Compliance with the specifications given in notes is as much a requirement as is meeting the dimensions on the part. Failure of the manufacturer to meet any noted requirement is grounds for part rejection. For this reason, the manufacturer must follow the notes, and the designer must not overspecify the requirements.

What information is given in a note depends partly on the use of the product definition documents. If the product definition is only used within the designing company, notes may make reference to standard company processes. Product definition used outside the designing company generally does not reference any of the designing company's standard processes. Instead, the documents describe procedures completely, or reference federal standards and industrial standards. Typical notes for installing fasteners are:

For use by the design company:

INSTALL FASTENERS PER SP663

For use by other companies:

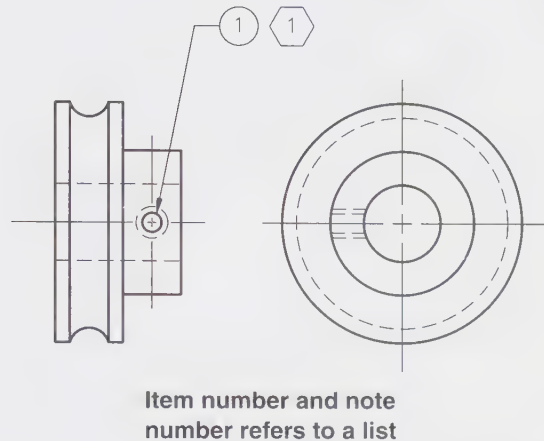
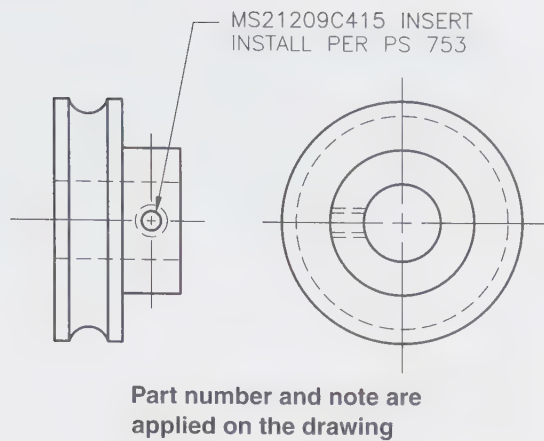
INSTALL FASTENERS PER DETAILED VIEW
SHOWING CORRECT FASTENER STACKUP.
TORQUE FASTENERS TO THE VALUES
SHOWN IN TABLE ONE.

The first note references a standard process manual that shows correct fastener installation and torque values. The second note gives all required information. Its use requires detail views showing fastener stackup and a table of torque values within the drawing.

Specific Notes

Information that applies to a specific feature is given in a specific note. See [Figure 3-37](#). A specific note may be shown on the drawing and connected to the applicable feature by a leader. If a separate notes list is being used, a note number is connected to the feature by a leader. The note number refers to a note in the list. Note numbers are placed inside a symbol that identifies it as a note number. The symbol must be different from the one used to identify item numbers on an assembly drawing. Use of one symbol for both purposes would result in confusion.

Two methods for showing a note callout that defines how to install a thread insert is shown in the given example. The first method shows the noted part number and installation process attached by a leader that points to the insert. The second method shows an item number and note number in place of the noted part number and process. The item number identifies the insert in a parts list, and the note number identifies the specific note that is applicable to the insert. The referenced note states the same requirement as shown in the previous example. The note is located in a list of notes that may be either on the field of the drawing or in a separate notes document.



Goodheart-Willcox Publisher

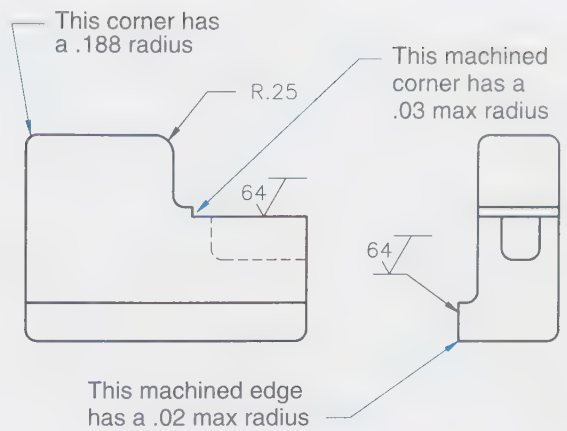
Figure 3-37. A specific note shows information applicable to a specific feature.

Application of the same note to several features is simplified by using note numbers. The note is only shown one time in the notes list, and the number is applied to each applicable feature. Drawing space and time are saved through this procedure.

General Notes

Information that applies to an entire drawing or model is contained in a general note. See [Figure 3-38](#). If a drawing is a mono-detail drawing (shows one part), then information that affects the entire part is a general note. If a drawing is a multi-detail drawing (shows more than one part), information that affects all parts on the drawing is a general note. Multi-detail drawings were fairly common in the past, but as CAD models replace the drawing method of documentation, the mono-detail format is becoming prevalent.

A fillet radius that is the same for most corners on a part may be given in a general note. The note specifies the radius one time and avoids the need for repeated radius dimensions on the part.



NOTES:

1. ALL FILLETS & ROUNDS R.188 UNLESS SPECIFIED OTHERWISE
2. ALL MACHINE CUTTER CORNER RADII .03 MAX UNLESS OTHERWISE SPECIFIED
3. REMOVE ALL SHARP EDGES AND BURRS PER SP 750 R.02 MAX
4. INFORMATION IN PARENTHESES IS REFERENCE INFORMATION

Goodheart-Willcox Publisher

Figure 3-38. General notes affect the entire drawing.

Only radii that differ from the noted radius must be dimensioned. A standard machining process for removing sharp edges and burrs is a common general note.

Pro Tip

General Notes

A general note is very precise in its requirements. It is not general in the sense of being vague. Vague notes are not of any use in product definition documentation. General notes are only general in the sense that they apply to the entire part or assembly in the drawing or model.

Separate Notes and Parts Lists

Placing notes and parts data on separate notes and parts lists allows the drawing sheets to be used entirely for the views, title block, and revision block. Information in a separate notes and parts list is considered part of the drawing. When drawings are manually prepared, it is common for preprinted forms to be used for drawing notes and parts list data. For CAD-generated drawings and models, starter files commonly called templates are used for notes and parts list information.

Some CAD systems automatically generate a notes list and parts list based on data entered as the design model is produced. The manner in which the notes and parts data is output from the CAD system depends on the particular software being utilized. It may be included on the field of the drawing or entered on a separate drawing sheet, or created as a separate file that is referenced by the drawing.

The use of standard templates saves time when creating separate notes and parts lists. There may be multiple templates used within one separate list.

A *cover sheet* is commonly used to show general information. It is normally an 8.5" × 11" sheet when printed. It shows the drawing number, signatures or electronic approvals, and the drawing title. Other information may be shown, such as the top assembly drawing number, model numbers, and serial numbers. The sheet number and total number of pages in the list are shown, such as 1 of 6. The drawing number and title are required to associate the list with the correct drawing. The originator's signature and an authorizing or approving signature are required by most companies.

Note sheets are used for the notes. A column for note numbers is provided. A space for the sheet number is also provided. A symbol of the company's choosing is placed at the beginning of each specific note to indicate that the corresponding note number is called out on the drawing or in the parts list. The presence of the symbol provides notice that the note number is shown on the field of the drawing where that note is applicable. The symbol used in the notes list can be a reduced version of the symbol used to enclose note numbers on the drawing. This symbol is not placed adjacent to general notes.

Parts list sheets are used to show information about parts and materials to be used in producing the parts or assembly. Spaces for the drawing number and sheet number are provided. Columns and headings are typically included in the templates.

One note should be placed above the title block or within the CAD file when a separate notes and parts list is used. The note indicates the existence of a separate list. This ensures that anyone looking at the drawing sheets or the model will know that a separate list is part of the documentation. For example:

SEE SEPARATE PARTS LIST

Notes on the Drawing

When a notes list is shown on the field of a drawing, all notes are placed in one area. Common locations are either the right or left margin of the first sheet. The first sheet may be used entirely for notes if there is a large amount of information. The notes list is headed by the word NOTES. The notes are numbered consecutively to make each one easy to locate.

Abbreviations

Abbreviations are used to reduce the space required for dimensioning and notes. Standard abbreviations for technical terms are defined in ASME Y14.38. Only standard abbreviations are to be used on a drawing. The use of nonstandard abbreviations is confusing and may result in production errors. An example of the space saved by abbreviations can be seen in the following example.

Abbreviated example:

2X .250-20UNC-3B THD X .38 DP

Non-abbreviated example:

.250-20 UNIFIED NATIONAL COARSE-3B
THREAD X .38 DEEP 2 PLACES

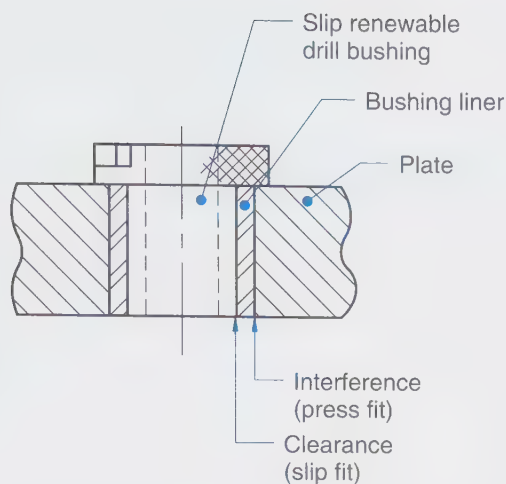
Acronyms are also used to reduce the space required for notes. An acronym is a series of letters made from the beginnings of the words in a phrase. The acronym does not always look like a word. An acronym may be created for any term or phrase that is used repeatedly in a notes list or report. However, it is better to use standard abbreviations and acronyms than to create one that may not be known or use one that is already standardized for a different term. The first time an acronym is used, the term or phrase is spelled out and the acronym is shown in parentheses. This provides the definition of the acronym for the reader. Any later use of the term only requires the acronym be shown. For example:

The Laser Altitude Sensor (LAS) must be assembled in a class 100,000 clean room.

After assembly, seal the LAS in its shipping container (P/N 11304).

Categories of Fit between Parts

The relationship between mated parts falls into two major categories: clearance fits and interference fits. See [Figure 3-39](#). A slip renewable drill bushing has a clearance fit that permits easy installation and removal. A drill bushing liner has



Goodheart-Willcox Publisher

Figure 3-39. Clearance fits permit parts to move, and interference fits hold parts in place.

an interference fit that provides a semipermanent installation.

Clearance Fit

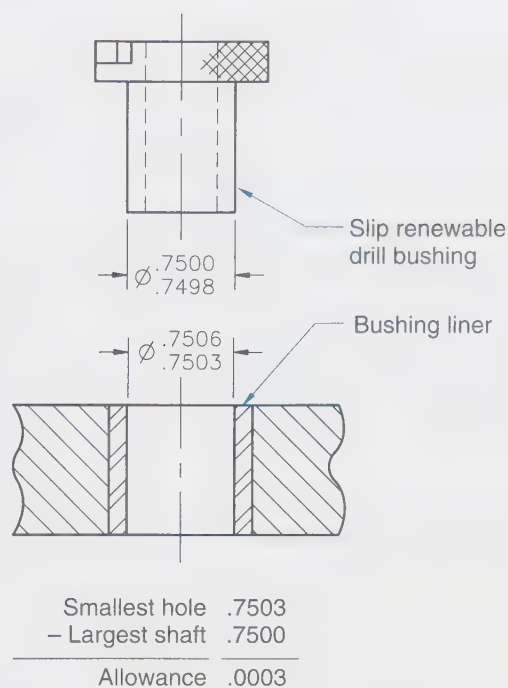
A *clearance fit* exists when an internal part is smaller than the mating external part. Clearance fits are used for moving parts. The amount of clearance provided between the moving parts is based on the amount of movement, rate of movement, finish on the parts, lubrication requirements, nominal feature size, operating temperatures, and loads.

Clearance fits are also used for location of parts relative to one another. See **Figure 3-40**. The figure shows a slip renewable drill bushing that slips into a drill bushing liner. The renewable bushing must clear the liner for easy installation and removal. It must also fit closely enough that the bushing location is accurately established by the drill bushing liner. The maximum amount of clearance is based on the amount of location error that can be tolerated between the bushing liner and drill bushing. The minimum clearance is based on the amount of clearance necessary to ensure removal of the drill bushing.

Interference Fit

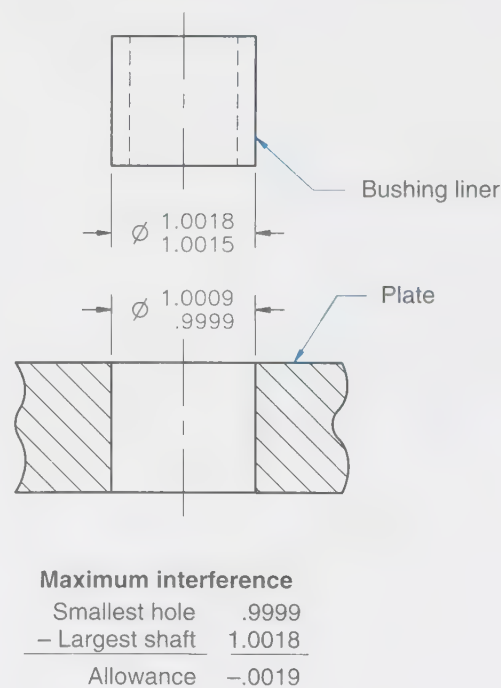
An *interference fit* exists when an internal part is larger than the mating external part. See **Figure 3-41**. The amount of interference depends on the function of the interference fit. A small interference is all that is required for most location interference fits. A larger amount of interference is required if the fit must hold parts together.

Too much interference between parts can cause damage. Hollow parts pressed into a hole



Goodheart-Willcox Publisher

Figure 3-40. Proper calculation of a clearance fit will result in limits of size that provide clearance between the features for all possible size combinations.



Goodheart-Willcox Publisher

Figure 3-41. A negative allowance value between two features indicates that an interference fit exists.

can collapse, and thin external parts may crack or rupture. Care must be taken to use only the amount of interference that is required for the design function. The stresses developed in an interference fit can be calculated using formulas contained in most engineering handbooks.

The bushing liner shown in the given example is pressed into the drill template (plate). The interference fit holds the liner in the plate and at the same time locates the liner. The amount of interference required for this design is relatively small because no substantial forces are applied to the liner.

Transition Fit

A *transition fit* permits size variations that can result in either an interference or clearance between two mating parts. See [Figure 3-42](#). Interference exists when the smallest permissible hole is mated with the largest permissible shaft. Clearance exists when the largest permissible hole is mated with the smallest permissible shaft.

Line Fit

A *line fit* has a zero allowance between mating parts. See [Figure 3-43](#). A *zero allowance* means that the difference between the maximum shaft and the minimum hole is equal to zero. A zero allowance is assumed to create an interference fit, because two parts with the same diameter do not have a clearance. In the given figure, an

interference fit only exists when both parts are at the .250" diameter. All other conditions for these two parts result in a clearance fit.

Geometric Tolerances

Control of form, orientation, location, runout, and profile are all achieved through the specification of geometric tolerances. Some of the symbols in Chapter 2 are used for geometric tolerances, and the topics in following chapters define how geometric tolerances are applied to drawings and models. The application of dimensions as defined in this chapter provides the size and location specifications from which the tolerance controls mentioned above can be specified.

Basic Dimensions

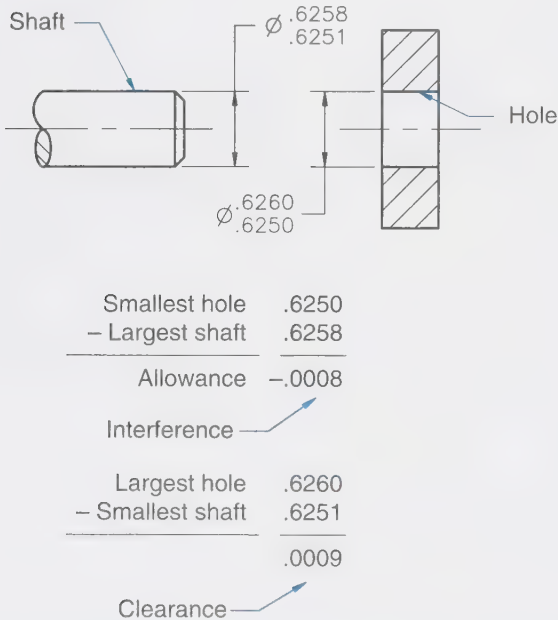
A *basic dimension* is a theoretical exact dimension value applied to the size or location of a feature. Title block tolerances or plus-or-minus tolerances specified in notes do not apply to basic dimensions.

A basic dimension is identified in one of two ways. A common method for identifying a basic dimension on a drawing or in a model is to place a rectangle around the dimension value. See [Figure 3-44](#). Another means for identifying basic dimensions is to use a general note. A typical note used to indicate basic dimensions is:

ALL UNTOLERANCED DIMENSIONS ARE BASIC

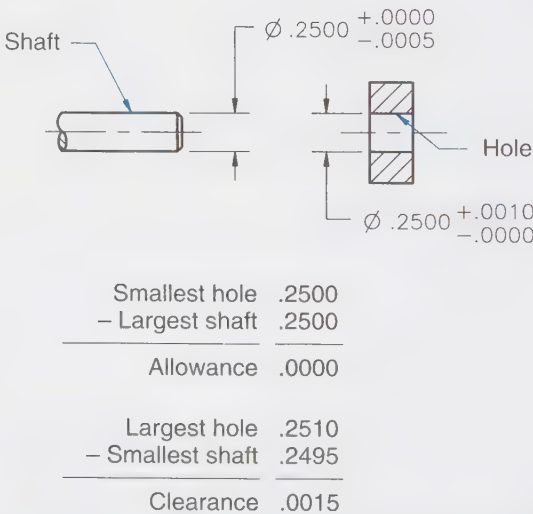
or

CAD DATA IS BASIC UNLESS OTHERWISE SPECIFIED



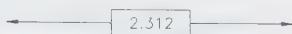
Goodheart-Willcox Publisher

Figure 3-42. A transition fit results in a clearance fit at one extreme of the applied tolerance limits, and an interference fit at the other extreme.



Goodheart-Willcox Publisher

Figure 3-43. A line fit results in line-to-line contact between mating parts. In this example, a line fit results when both parts are at the .250" diameter.



Goodheart-Willcox Publisher

Figure 3-44. A basic dimension is identified by placing the dimension value inside a rectangle.

When a general note is used on a dimensioned drawing, each dimension is considered basic unless it has a specific tolerance associated with the dimension value. A dimension of 2.500" is basic if a general note is used. A dimension of 2.500" \pm .008" is not basic whether or not a general note is used. When undimensioned CAD models are used, a general note states that the CAD data is basic, thus making all queried values basic.

A past practice for indicating basic dimensions was to place the word BASIC or the abbreviation BSC adjacent to the dimension value. This method is no longer defined in the dimensioning standard.

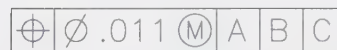
The purpose for basic dimensions is to provide the theoretical values for feature sizes and locations from which geometric tolerances are specified. Any feature size or location defined by basic dimensions will normally have a tolerance applied through the application of a feature control frame.

Feature Control Frames

A *feature control frame* is used to specify the tolerance that applies to specific features or patterns of features. See [Figure 3-45](#). Feature control frames always contain at least a tolerance symbol (characteristic) to indicate the type of control and a tolerance value to indicate the amount of acceptable variation. The complete composition of feature control frames is defined in Chapter 2. Information on methods for applying these specifications to a drawing or model is briefly described in that chapter. The tolerance specifications that can be achieved through the use of feature control frames are explained in the following chapters.

Material and Boundary Condition Modifiers

Tolerances specified on features of size are applicable regardless of material condition, unless a modifier is shown. The modifier indicates at what material condition the tolerance is applicable. If the tolerance must be met at all possible



Goodheart-Willcox Publisher

Figure 3-45. A feature control frame contains tolerance requirements that apply to a feature or a pattern of features.

produced sizes of the feature, it is said to apply *regardless of feature size (RFS)* and no modifier is shown. Tolerances that must be met only when the most material exists in the part are said to apply at *maximum material condition (MMC)*. Tolerances that must be met only when the least material exists in the part are said to apply at the *least material condition (LMC)*.

Material condition modifiers are included in tolerance specifications to indicate at what material condition the tolerances apply. It is only necessary to show the MMC or LMC modifiers because RFS is assumed when no modifier is shown. See [Figure 3-46](#). When referencing material conditions within a note, the applicable term is typically spelled out or an abbreviation may be used.

Material boundary modifiers are included in datum feature references to indicate at what material boundary the datum is to be established. This will be further explained in the chapter on datums and datum feature references. Datum feature references default to be applicable regardless of material boundary, but it is also possible to apply a symbol indicating the datum feature reference is applicable at the maximum material boundary or at the least material boundary. The symbols used to specify applicable material conditions for tolerances (MMC and LMC) are also used to specify applicable boundary conditions for datum feature references (MMB and LMB). When invoking material boundaries within a note, the applicable term or abbreviation is typically used rather than applying a symbol, but symbols may be used.

- (M) Maximum material condition (MMC)
Maximum material boundary (MMB)
- (L) Least material condition (LMC)
Least material boundary (LMB)

Goodheart-Willcox Publisher

Figure 3-46. Material condition modifiers are applied to tolerances to indicate at what material condition the tolerance is applicable. Material boundary modifiers are applied to datum feature references.

Chapter Summary

- ✓ Size description is defined through dimensions.
- ✓ Location dimensions define where a feature is located.
- ✓ Size dimensions indicate how large a feature is.
- ✓ Extension lines are used to extend features for the application of dimensions.
- ✓ Break extension lines only where they cross an arrowhead or are sufficiently close to an arrowhead as to cause confusion.
- ✓ Dimensions are placed inside a break in the dimension line.
- ✓ The minimum recommended space from an object to a dimension line is .40" (10 mm). The minimum recommended space between dimension lines is .24" (6 mm).
- ✓ All dimensions on a drawing are based on one measurement system. Exceptions are identified by showing the units of measurement beside each dimension that is an exception.
- ✓ A combination of dimensioning systems may be used to control tolerance accumulation, and the best control of tolerance accumulation is achievable through the application of datums and geometric tolerances.
- ✓ Size and location for every feature on a part must be given.
- ✓ Dimensions are applied where the feature contour is visible.
- ✓ Dimensions are applied between views when practical. Dimensions are applied off the views (outside the objects).
- ✓ Dimensions are generally applied with consideration of how parts fit together.
- ✓ Large dimensions are generally placed outside small dimensions.
- ✓ Stagger dimension values to avoid confusion. Avoid dimensioning to hidden features.
- ✓ No feature should have two methods for defining its size. Do not double dimension.
- ✓ Avoid using pictorial drawings to define size for production.
- ✓ Notes are worded precisely to give exact information.
- ✓ Notes are part of the product definition, even when shown on a separate list. Notes define part requirements that must be met.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. Dimensions are used to specify _____.
A. size and volume
B. size and location
C. size and shape
D. shape and location
2. _____ lines are used to extend features for the application of dimensions.
A. Dimension
B. Leader
C. Extension
D. Object
3. An extension line may be broken when it crosses _____.
A. another extension line
B. an arrowhead
C. an object line
D. All of the above.
4. It is preferable to show dimension values and dimension lines _____.
A. outside the object outline
B. inside the object outline
C. Both A and B.
D. Neither A nor B.
5. Leader lines in an orthographic view should not be drawn _____.
A. horizontal
B. vertical
C. at a 45° angle
D. Both A and B.
6. The dimension values are oriented to be read from _____ when using the unidirectional dimensioning method.
A. the right or left side
B. the bottom and right side
C. the bottom
D. any convenient direction
7. The dimension values are oriented to be read from _____ when using aligned dimensioning in orthographic views.
A. the right or left side
B. the bottom and right side
C. the bottom
D. any convenient direction
8. When dimensioning in inch units, it is preferable to use _____ values.
A. full inch
B. fractional inch
C. decimal inch
D. Both B and C.

9. A zero is placed in front of dimension values less than 1 when using the ____ measurement system.
- metric
 - inch
 - pound
 - Both A and C.
10. The ____ dimensioning system uses a single feature as the origin for dimensions to all other features.
- chain
 - direct (combined)
 - baseline
 - Both A and C.
11. Polar coordinates include a(n) ____ dimension.
- angle
 - latitude
 - longitude
 - Both B and C.
12. Dimensions should be applied to the view in which the ____ of the feature is clearly shown.
- texture
 - centerline
 - contour
 - height
13. Placing dimensions ____ permits easier association of the dimension to multiple views.
- inside the object outline
 - outside the object outline
 - close to the object outline
 - between the views
14. In an orthographic view, dimension values in multiple adjacent vertical dimensions are usually ____ to make them easier to read.
- aligned with each other
 - centered
 - staggered
 - underlined
15. Features are generally dimensioned where they are ____.
- hidden
 - visible
 - Either A or B.
 - Neither A nor B.
16. A ____ dimension is provided for general information only, and is not a requirement that must be met.
- radius
 - reference
 - limit
 - polar

True/False

17. *True or False?* An extension line is not broken where it crosses another extension line.
18. *True or False?* A dimension line may be drawn with both arrowheads on the outside of the extension lines, and the dimension value shown between the extension lines.
19. *True or False?* When the note **UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS ARE IN INCHES** is used, any metric dimensions must have the abbreviation mm applied as a suffix.
20. *True or False?* The scale of a drawing determines the dimension value that is applied to a feature.
21. *True or False?* A metric dimension must show the abbreviation for millimeter even if all dimensions on the drawing are in millimeters.
22. *True or False?* An origin must be indicated by showing 0,0 values when using the rectangular coordinate dimensioning without dimension lines system.
23. *True or False?* Only aligned dimension values may be used without dimension lines in the rectangular coordinate dimensioning system.
24. *True or False?* Tabulated dimensions cannot be used to define dimensions for any features other than holes.
25. *True or False?* It is not always possible to place every dimension between views.
26. *True or False?* It is preferable to avoid extending a small dimension across a larger one.
27. *True or False?* It is permissible to dimension to a hidden line when the hidden line is part of a feature in a half section view.
28. *True or False?* It is preferable to dimension a part feature so that its size or location can be determined in more than one way.
29. *True or False?* Notes provide supplementary information and are not considered to be part of the drawing requirements.
30. *True or False?* A basic dimension indicates that title block plus-or-minus location tolerances do not apply. It also indicates that the tolerance on the feature is probably provided in a feature control frame.

Fill in the Blank

31. A(n) _____ is used to connect a notation to a specific feature on the drawing.
32. A dimension applied to a 6" feature that is drawn at half scale must show a dimension of _____ inches.
33. Arrowheads are typically drawn with a length-to-width ratio of _____.
34. The _____ dimensioning system places dimensions in-line, and is also known as the point-to-point dimensioning system.
35. The _____ dimensioning system shows size and location dimensions in a table.
36. _____ dimensioning results in dimensions that show two acceptable limits of size or location.
37. The origin symbol is a _____ drawn in place of one of the arrowheads on the dimension line.
38. Notes may be placed on the field of the drawing, but they are usually listed in a _____.

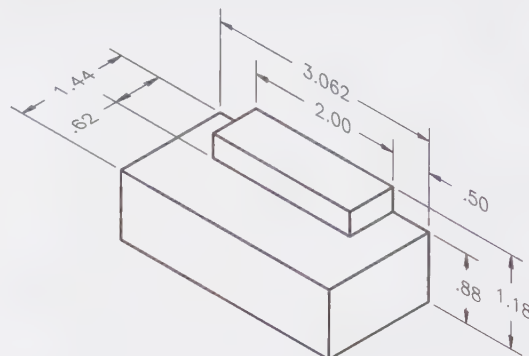
Short Answer

39. Define *unidirectional dimensions*.
40. Define *aligned dimensions*.
41. Explain the effect of using chain dimensions for all features along the length of a part.
42. Explain how tabulated dimensions are used to provide location and size requirements for holes.
43. If two parts in an assembly mount on the same surface, explain why the mounting surface should be used as a baseline (origin) for dimensioning each part.
44. Explain why larger dimensions are generally placed outside smaller ones.
45. Explain what is indicated by the origin symbol when it is applied to a drawing.
46. Explain what must be done to avoid dimensioning internal features where they appear as hidden lines.
47. Define *maximum material condition*.

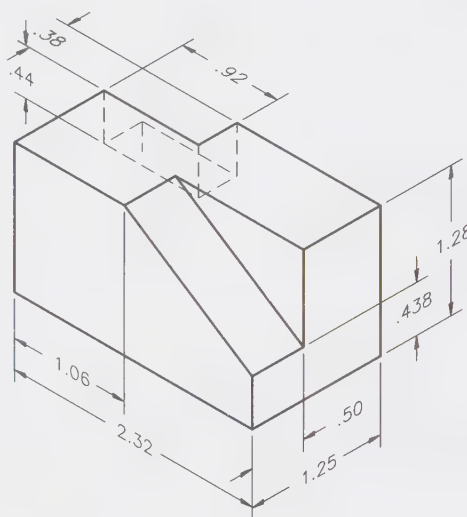
Application Problems

Each of the following problems shows a dimensioned pictorial drawing. Sketch the necessary orthographic views of each problem to permit application of all dimensions for the part. Apply dimensions to completely define size and location of all features. Dimensions are to be applied in compliance with guidelines covered in this and preceding chapters.

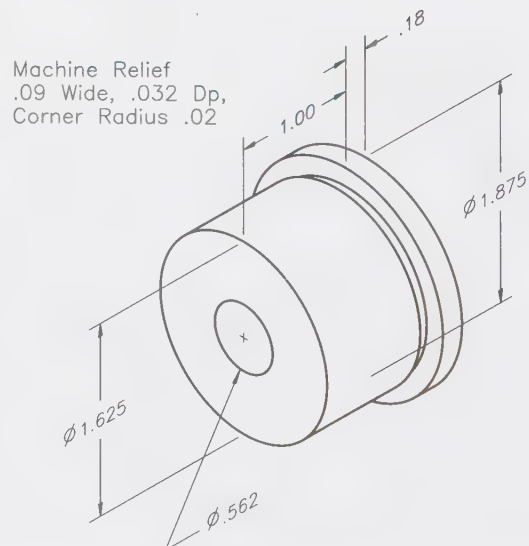
48.



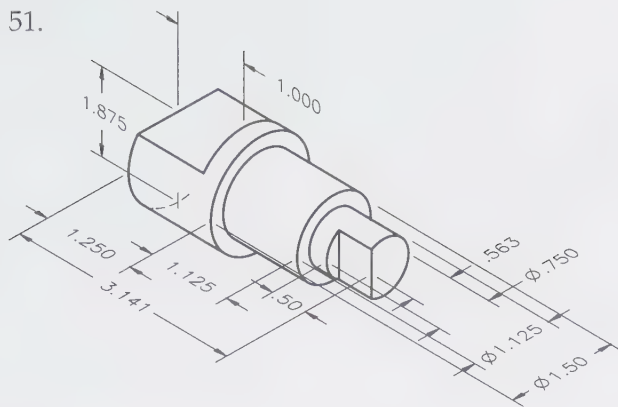
49.



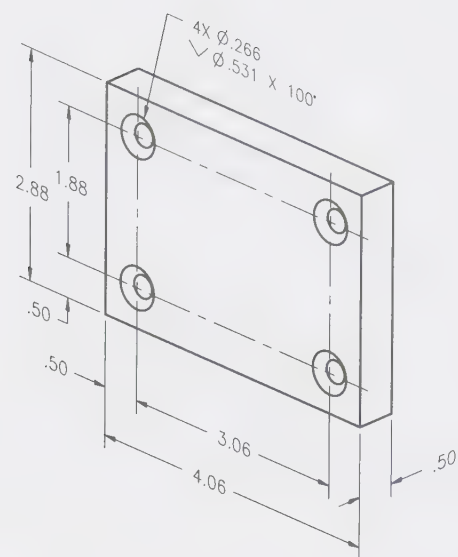
50.



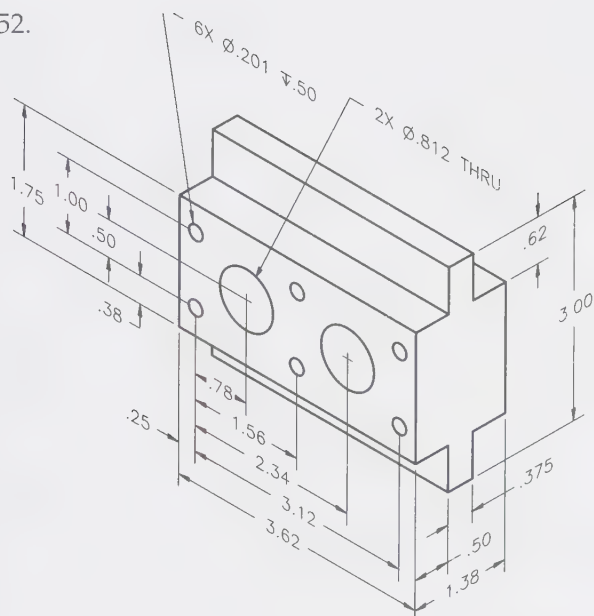
51.



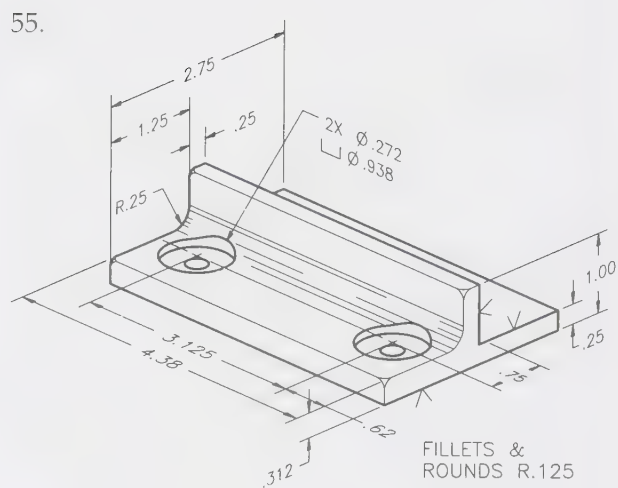
54.



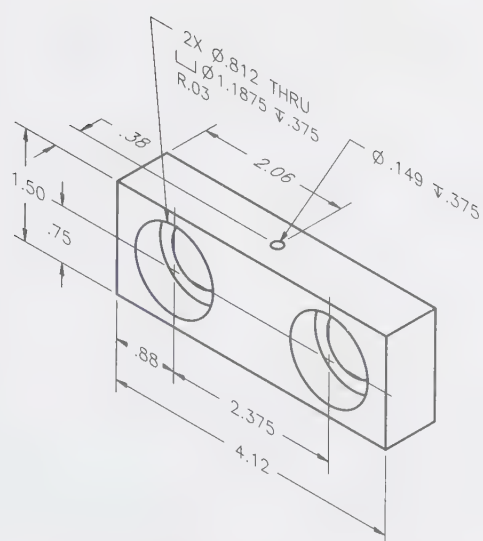
52.



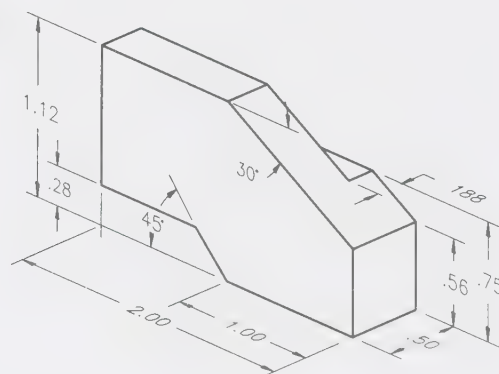
55.



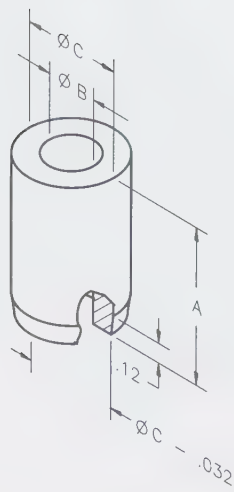
53.



56.

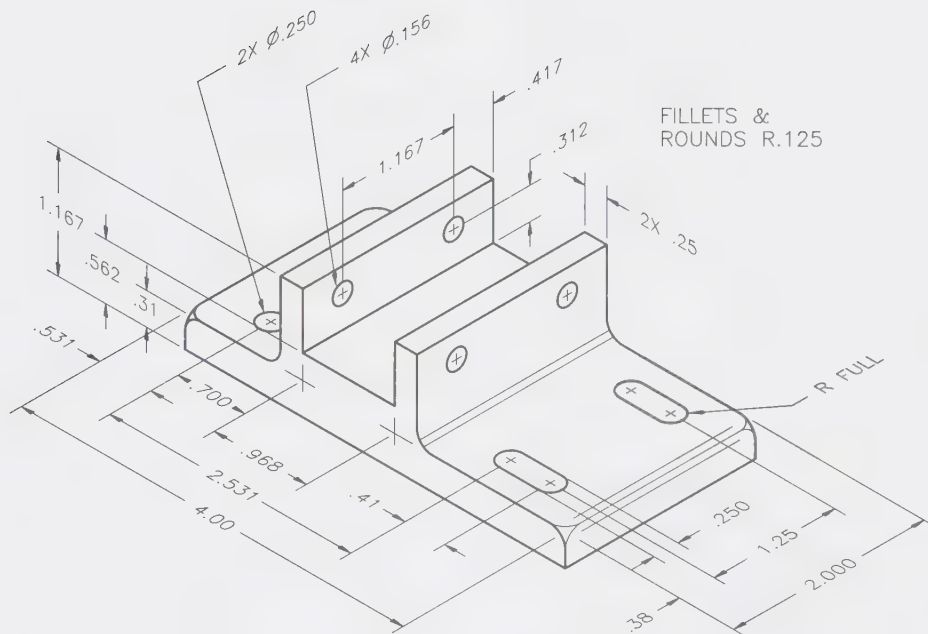


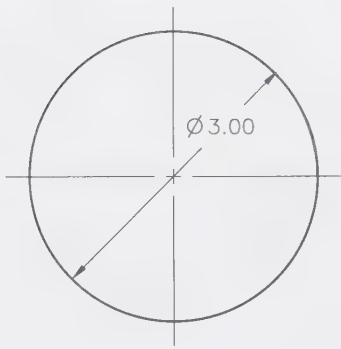
62.



Problem Number	A	B	C
62a	.750	.6250 $^{+.0001}_{-.0005}$.8768 .8765
62b	1.000	.3281 $^{+.0001}_{-.0005}$.6267 .6264

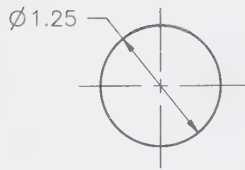
63.



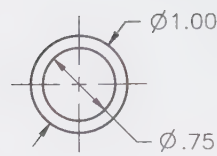


Dimension line and
size inside

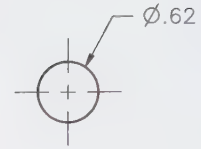
Large hole



Dimension line
inside, size outside

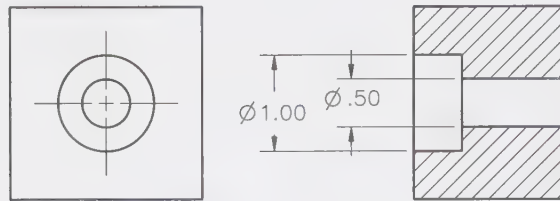


Combination of
application techniques



Application by
leader

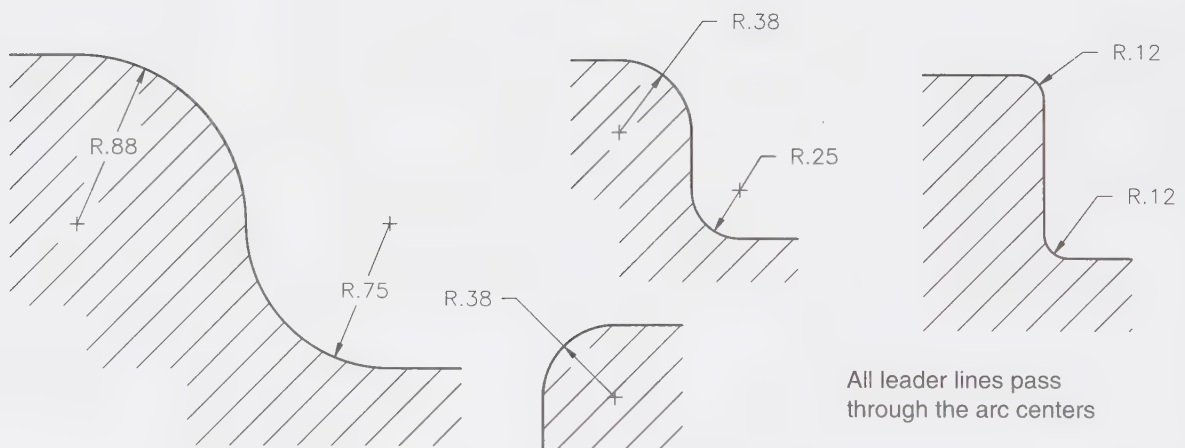
Small hole



Hole diameter dimensioned
on the noncircular view

Dimensioning hole diameter

Goodheart-Willcox Publisher



All leader lines pass
through the arc centers

Arc dimensions

Goodheart-Willcox Publisher

Chapter 4

Dimension Application and Limits of Size

Objectives

Information in this chapter will enable you to:

- ▼ Clearly apply dimensions by complying with the stated general dimensioning guidelines.
- ▼ Apply dimensions to any of the geometric shapes commonly found on mechanical parts.
- ▼ Cite the categories for limits of fit and describe the general condition created by each category.
- ▼ Calculate and apply limits of size for mating features.
- ▼ Explain Rule #1 and Rule #2 of the ASME Y14.5-2009 standard.
- ▼ Provide examples of the effects that dimensions and tolerances have on manufacturing.
- ▼ Complete a surface texture specification when provided the allowable variations.

Technical Terms

allowance
apex offset
arc tangents
base diameter
basic hole system
basic shaft system
bilateral tolerance
blind holes
centerdrill
cone height
counterbore
countersink
diameter
features of size
irregular features of size
joggle
keyseat

lay
limit dimensions
mate drilling
minor radius
nominal size
parallelepiped
regular features of size
removed view
revisions
right square pyramid
roughness
roughness cutoff
roughness width
roughness width cutoff
sampling length
sharp diameter
single limit dimensions
spotface
surfaces
tolerance
unilateral tolerance
waviness

Introduction

A general description of a part can be given through the standard views, section views, and auxiliary views of a multiview drawing or through pictorial views. The size, location, and values affecting geometric form requirements are given through the application of dimensions and tolerances. Complete part description including as-designed dimensions may be defined by CAD model geometry with tolerances applied to the model.

Complete part definition requires that all aspects of each geometric feature on a part, including dimension and tolerance requirements, be

defined. Each part can be visualized as a composite of geometric shapes, and each of those shapes dimensioned and toleranced. Methods for applying dimensions on various geometric shapes are contained in this chapter. Application of geometric tolerances is defined in following chapters.

Features fall into two general categories: *surfaces* and *features of size*. There are two classes for features of size. *Regular features of size* are typically associated with a single size dimension such as thickness, width, length, or diameter. *Irregular features of size* may be one enclosed shape or a collection of features that in combination establish a size boundary. Throughout this text, the term *feature of size* is general and intended to encompass both the regular and irregular features of size.

Each size dimension must have an allowable amount of acceptable variation, because it is not possible to build multiple parts that are all the same size. There is a standardized means for showing size limits on dimensions applied to features of size. Those size limits define the amount of allowable variation that may occur in fabrication of the features. Allowable variations are calculated on the basis of design function. For standardized fits, the calculations are completed using standard tolerance tables.

Dimension Application

Specific dimension application techniques are used for common geometric shapes. The use of a specific application technique for each feature type provides a consistent and recognizable dimensioning system. Consistency in the application techniques prevents misinterpretations that might occur if random methods were used. A consistent set of application techniques ensures that dimensions are easy to interpret because the techniques are repeated every time the same geometric shape is repeated.

The dimension application techniques described in this section were established for dimensioning orthographic drawings. These practices may also be utilized for orthographic drawings created from three-dimensional computer-aided design (CAD) models. However, these practices are not always applicable in three-dimensional models. The CAD geometry itself may be queried to determine dimension values instead of creating and displaying the dimensions. Using the geometry to determine dimensions does not by itself establish the desired relationships between specific features. For this reason, undimensioned CAD

models should always include datums and geometric tolerances to ensure that the correct relationships between features are specified.

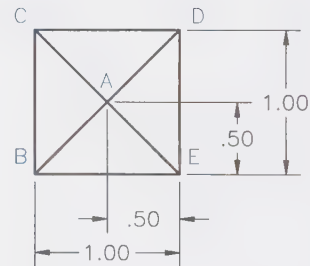
Prisms and Pyramids

Prisms and pyramids have edges where the adjacent sides intersect. See [Figure 4-1](#). The edges intersect to create corners. Dimensions that show the locations for all the corners describe the size of the object.

The given right rectangular *parallelepiped* (prism) has eight corners. The corners are at the intersections of the edges created by the six flat surfaces that form the prism. Because opposite sides of this particular type of prism are known to be parallel, it is only necessary to show three dimensions. Three dimensions completely describe the



Prism



Pyramid

Goodheart-Willcox Publisher

Figure 4-1. Height, width, and depth dimensions must be defined for prisms and pyramids.

size of the object by showing its height, width, and depth. In orthographic views, the angle between surfaces is assumed to be 90° because the object appears as a parallelepiped. It is not necessary to dimension 90° angles in an orthographic view. In a model, the 90° angle is not assumed but may be determined by querying two sides of the part to determine the angle.

The given pyramid is a *right square pyramid*. It has five points, all of which must be dimensioned. The apex (point A) is dimensioned from the base (surface B,C,D,E) at a height of 1.25". Width and depth dimensions for the base are each given as 1.00". Point A is shown centered above base B,C,D,E by two locating dimensions of .50".

The .50" dimensions provide the distances as seen in the view where the dimensions are applied. The .50" dimensions specify rectangular coordinates relative to the two edges and not the true distance from the edges to the apex. The 1.25" height dimension must also be considered if the true distance from the base edges to point A is desired. The dimensions shown are the correct dimensions for specifying the size of the given pyramid. All point locations are dimensioned, and the part can be produced when all point locations are known.

Cylinders

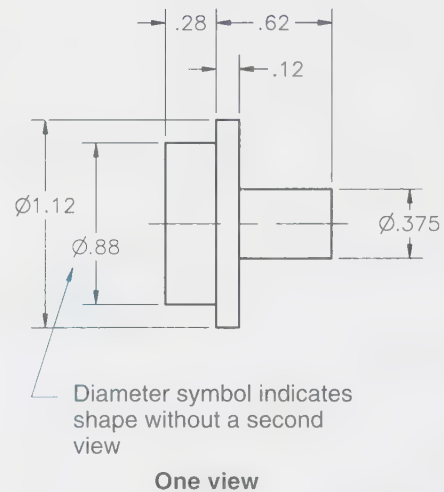
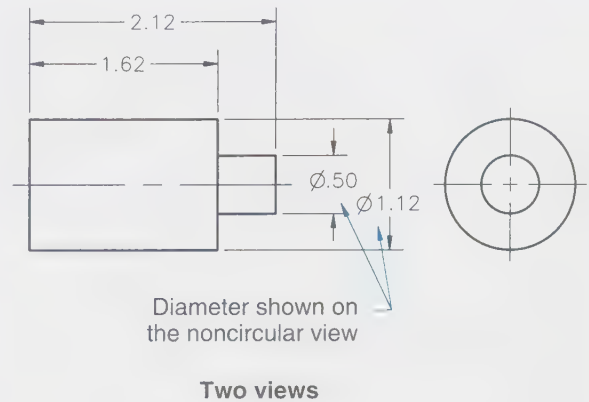
The diameter and length must be given for a cylinder. It is preferable for both the diameter and length to be dimensioned on the noncircular view. See [Figure 4-2](#). The diameter dimension is normally placed between the noncircular view and the circular view. Cylinders are dimensioned by specifying the diameter rather than the radius.

One view may be used to show a cylindrical object if all features can be clearly dimensioned, as shown in the second example. The diameter and length for each cylinder is specified; no additional dimensions are required. A diameter symbol is placed in front of diameter dimensions. The diameter symbol is a circle with a diagonal line through it.

Some situations require that the diameter dimension for a cylinder be shown in a circular view. See [Figure 4-3](#). The given cylinder has a flat cut down its length, and the noncircular view does not show the full diameter. Dimensioning the .75" diameter in the noncircular view would be confusing.

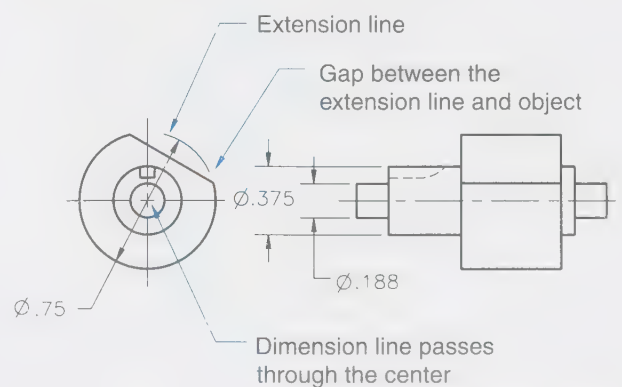
On the circular view, an extension line is drawn using an arc that is equal in radius to the one for the cylinder. The extension line is drawn thin like all extension lines. A gap is left between the object and the extension line. The extension line runs about .125" past the dimension line. The dimension line passes through the center of the

cylinder. One end terminates on the cylinder, and the other end extends outside the cylinder to a position where the dimension value is shown. A short horizontal line extends between the dimension line and the dimension value. The diameter



Goodheart-Willcox Publisher

Figure 4-2. The diameter and length must be shown for a cylinder.



Goodheart-Willcox Publisher

Figure 4-3. In some cases, it is necessary to dimension the diameter of a cylinder in the circular view.

symbol is required to avoid the possibility of the dimension being interpreted as a radius. Arrowheads are drawn at the cylinder and extension line.

Flats on Cylinders

A flat is often cut into a cylinder to provide a surface suitable for clamping something in a fixed location. The size of a flat on a cylinder is dimensioned as shown in **Figure 4-4**. The location of flat A,B,C,D is specified relative to the far side of the cylinder. The flat's location may also be specified relative to the centerline or other features on the cylinder, when any other features exist. Dimensions from centerlines can be ambiguous without datum feature references or when multiple coaxial features exist. If dimensions are applied from the centerline, a datum axis should be established and a profile tolerance referencing that datum axis applied to the flat.

The length of the flat is shown in a noncircular view. Locations of edges A,B and C,D are determined by the cylinder's diameter and the flat location dimension. The locations of edges A,B and C,D are not dimensioned directly.

The choice of which location dimension to use on a flat is based upon the part's function. If the flat is a clamping surface, a dimension from

the far side is preferred. This dimension is easy to inspect and it is an acceptable method when large errors relative to the cylinder centerline can be tolerated. If the flat is a locating surface for part of a mechanism, its location to the cylinder's centerline may be specified. This makes inspection more difficult because the centerline must be established from the cylinder's diameter, but the function of the part usually takes precedence over the ease of inspection.

Cones

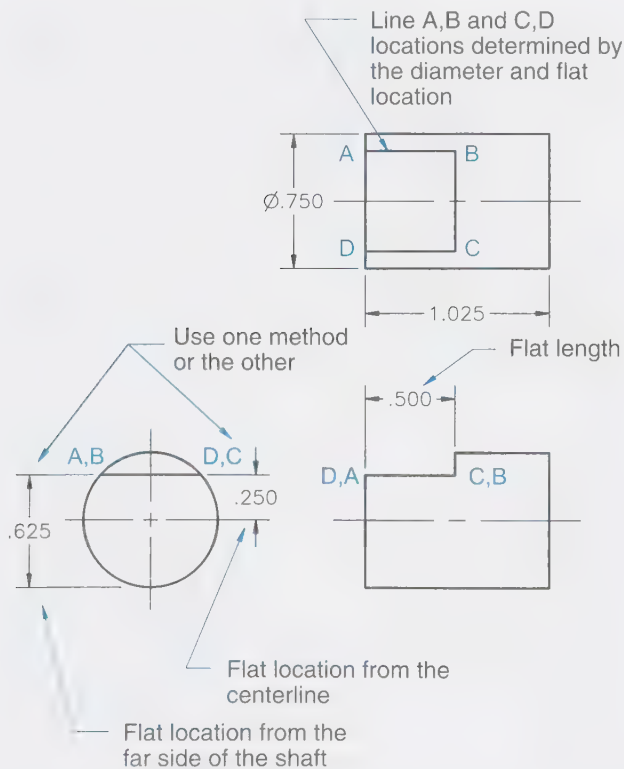
A right circular cone is normally dimensioned by giving the **base diameter** and the **cone height**. See **Figure 4-5**. It is also acceptable to show the base diameter and the cone angle. An oblique circular cone is dimensioned by giving the base diameter, cone height, and **apex offset**.

Holes

Holes are normally dimensioned by giving the **diameter**. The dimensions may be shown in the views where the holes appear as circles. See **Figure 4-6**. Large diameter holes are dimensioned by placing the dimension line and size value inside the hole. The dimension line and value are placed inside the hole if the hole is large enough. If the diameter is too small for the value to be shown on the inside, the dimension line and arrowheads are placed inside and the value is placed outside of the hole. The dimension line must pass through the center of the hole. The technique of placing the dimension value outside the circle is also used when centerlines and concentric circles interfere with the dimension value.

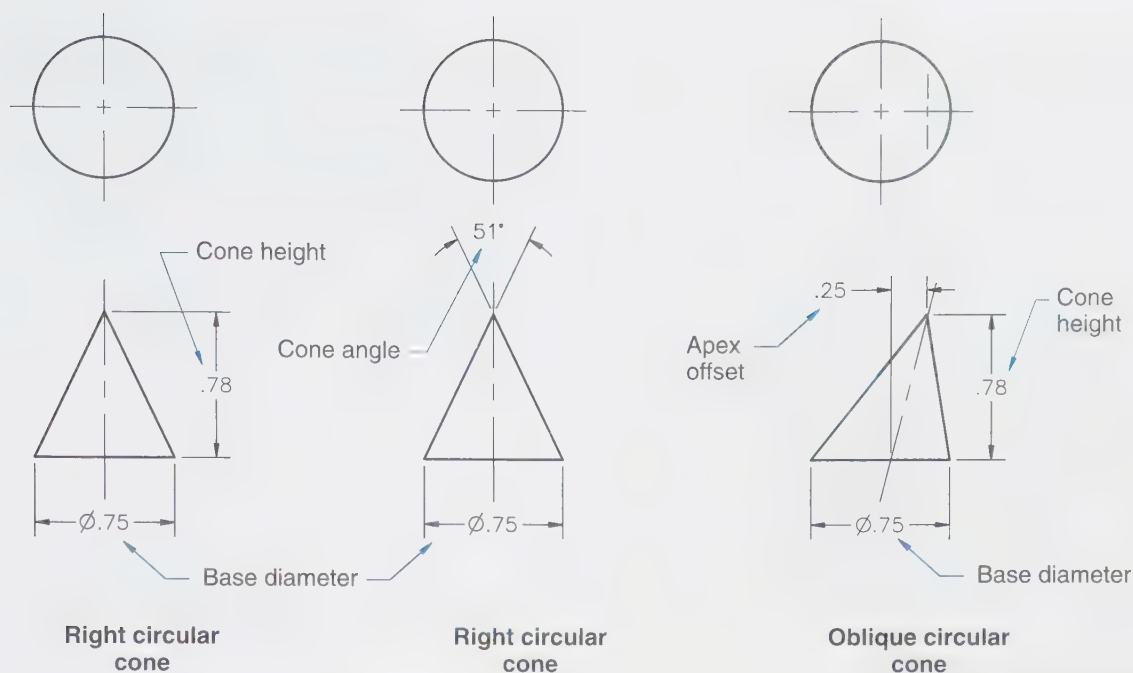
Concentric diameters may be dimensioned using a combination of techniques. The .75" diameter hole in the given example is dimensioned by placing the dimension line on the inside and the value on the outside. The 1.00" diameter counterbore is dimensioned by placing the dimension line and value on the outside. This combination of dimensions can be used to prevent the dimensions from being crowded. Other combinations may be used provided that each diameter is clearly dimensioned.

Small holes are dimensioned with a leader that connects the specified size value to the hole. The leader may include a horizontal line extending from the dimension value. If a horizontal line is used, a distinct corner is made at the end of the horizontal line, and the leader extends from the corner toward the center of the hole. The leader is terminated with an arrowhead where the leader intersects the hole.



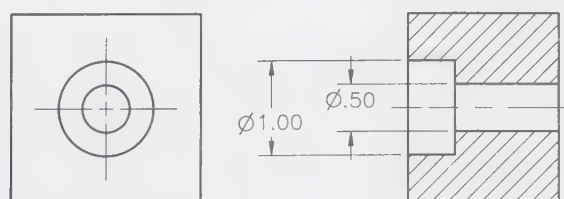
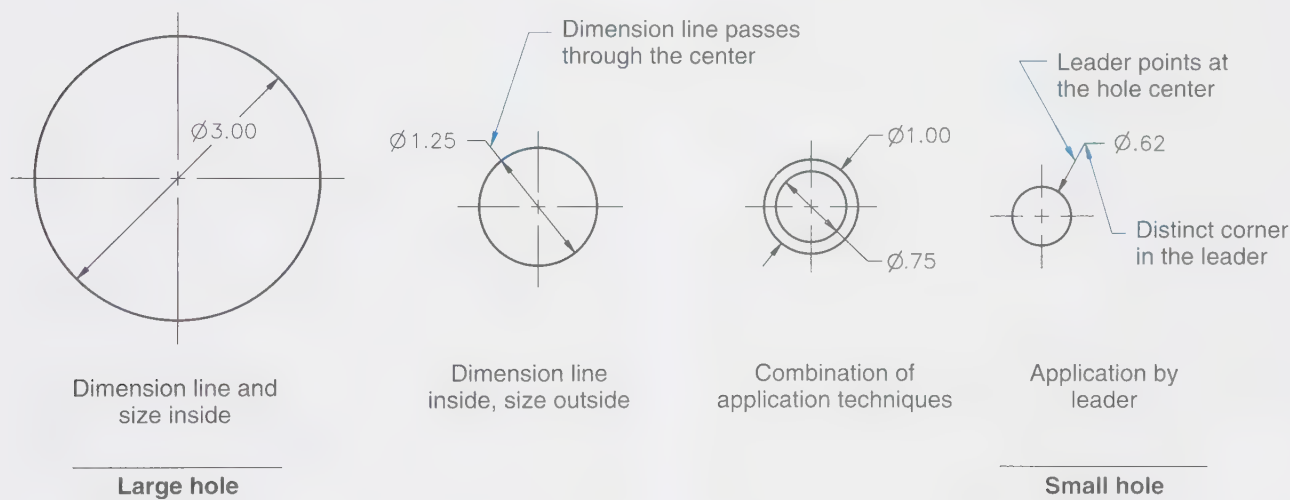
Goodheart-Willcox Publisher

Figure 4-4. The location and length of a flat must be dimensioned on a cylinder.



Goodheart-Willcox Publisher

Figure 4-5. The diameter of the base, cone height, and any required apex offset are given when dimensioning a cone.



Hole diameter dimensioned on the noncircular view

Dimensioning hole diameter

Goodheart-Willcox Publisher

Figure 4-6. The diameter must be specified for holes.

Hole diameters may be specified in a noncircular view if the hole is shown with object lines. This is only possible when the noncircular view appears in a section. Hidden noncircular views are not normally dimensioned.

The callout of a hole size does not require that a process for making the hole be specified. If a hole of $.250 \pm .005$ " diameter is specified, the fabricator or production planner must decide how to achieve the desired size. A fabrication process may be specified along with the dimension when a specific process is necessary to meet the design requirements. As an example, for some structural applications (such as aircraft parts) it may be required that a hole in sheet metal be drilled, reamed, or bored rather than punched.

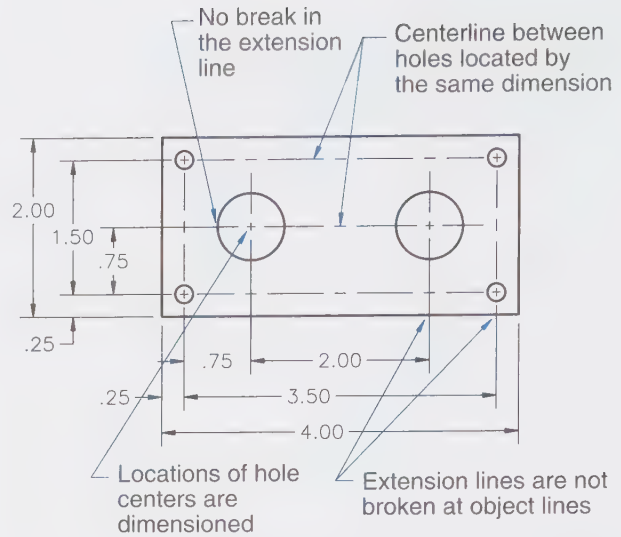
The use of specified hole sizes that correspond to standard tool diameters is a good practice. A size tolerance is also required for holes. It is important to realize that the magnitude of the tolerance specified does impact the process selected for producing the hole. If tolerances of $+.006$ " and $-.001$ " are specified, the tolerance is large enough that most holes under $.500$ " diameter can be drilled, which is a relatively inexpensive process.

When tolerances too small to permit drilling are used, it may be necessary to use more expensive processes to produce the holes. As an example, a small hole diameter tolerance can be achieved by first drilling the hole undersize, then finishing the hole by reaming it to the specified size.

As hole size tolerances are decreased, the amount of precision and time taken to produce them increases. When tolerances under $.001$ " are specified, the hole may need to be bored, ground, or lapped to meet the specification. These are expensive processes compared to casting, molding, punching, drilling, or reaming. Hole diameters and tolerances must be specified to achieve the design goals, but specification of tighter-than-necessary tolerances must be avoided to keep manufacturing costs down.

A hole location is dimensioned to the hole's center. See [Figure 4-7](#). This is done because hole-cutting tools rotate on a machine spindle, and the location of the machine spindle center can be accurately located. Some processes such as laser or water jet cutting may profile the hole, but the hole location is still dimensioned to the hole center.

Extension lines project from the hole center to the dimension lines. Typically, no breaks are shown in an extension line once it is outside the hole, but current drawing practices do permit the utilization of centerlines in place of extension lines. Centerlines are drawn between holes when



Goodheart-Willcox Publisher

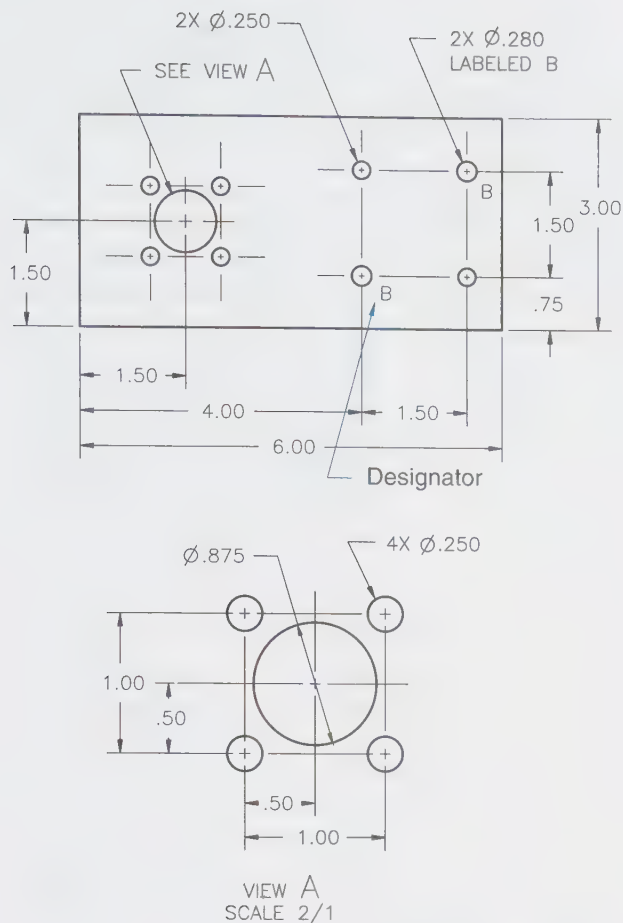
Figure 4-7. Hole locations must be dimensioned to the centers.

a single dimension is used to locate multiple holes that are aligned with one another.

Hole sizes and locations must be dimensioned unless CAD models are used to define the geometry requirements. Locating dimensions are placed on one side of the view, if possible, permitting hole size specifications to be placed on the other side of the view. This arrangement prevents leaders from crossing dimension lines and makes the drawing easier to read. As parts become complex, it may not be possible to always stay in compliance with this general guideline.

A simple plate for which all holes are completely dimensioned is shown in [Figure 4-8](#). A hole pattern is located on the plate, and dimensions for the hole pattern are shown in removed VIEW A. A *removed view* is used to reduce crossed dimensions on the drawing and to provide a clear definition of small features. The center of the $.875$ " diameter hole is used as an origin in VIEW A. The location of the $.875$ " diameter hole is given in the main view. The four $.250$ " diameter holes are located from the center of the $.875$ " diameter hole.

The main view includes location dimensions for four holes in addition to those shown in VIEW A. Two holes are $.250$ " diameter and two are $.280$ " diameter. A letter **B** is used to designate the two $.280$ " diameter holes. The designator is used because there is only a $.030$ " difference in diameter between the two hole sizes. The small difference in diameter makes it difficult to determine which holes are of what diameter without the designator. The designator is placed in the same position relative to each identified hole. This ensures association of the correct hole with its designator.



Goodheart-Willcox Publisher

Figure 4-8. Complex features or patterns of features may be dimensioned in removed views.

The hole size specification shows the number of holes to be made at each specified size. The standard method first shows the number of times a hole is repeated and then shows the diameter. The 2X indicates the hole occurs two times. The X symbol has a meaning of *times* when no space is between it and the preceding number.

Example:

2X Ø.250

(two places, or instances, diameter .250")

A past practice is to write out the number of holes as shown in the following example. This method is no longer illustrated in the standard.

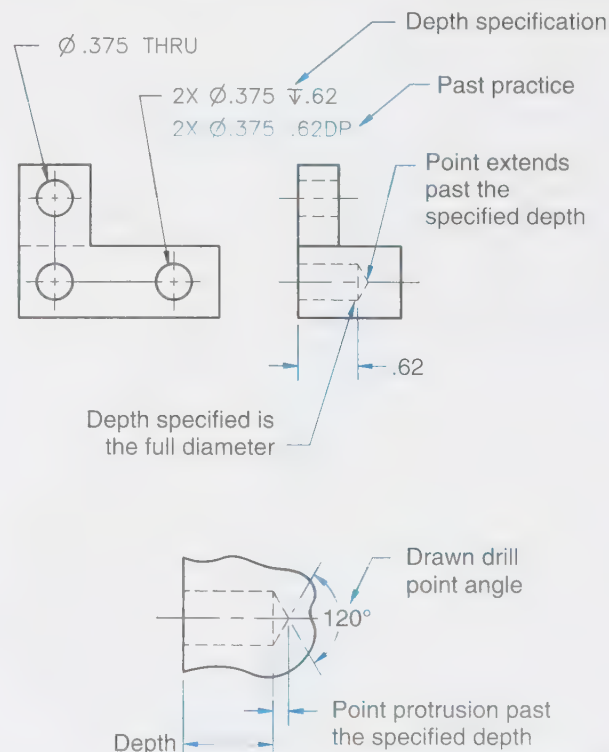
Ø.250 2 HOLES

Holes are assumed to go through the part on which they are shown unless a depth is specified or defined by a model. See **Figure 4-9**. If the drawing views do not make it clear that a hole goes through the part, the word **THRU** must follow the hole diameter. Depth must be specified for any hole that does not go through a part. Holes that do not go through are called **blind holes**.

The depth of a hole is specified following the diameter. The depth symbol is placed in front of the dimension value. The depth is the distance that the full diameter of the hole extends into the part. The drill point extends beyond the specified depth for a hole.

Depth of the full diameter is specified because it is the usable portion of the hole. The point protrusion past the full diameter must be considered and steps taken to ensure that the point does not unintentionally break out the far side of a part. Drill points when manually drawn are shown at an included angle of 120°. Actual drill point angles and shapes vary. However, 118° is fairly standard; thus, a 120° included angle is sufficiently accurate. When using CAD, the solid model may include the true point shape of the drill. Whether or not the actual drill shape is modeled may be dependent on internal company standards.

Hole depth is generally not more than three to five times the diameter of the hole. This allows the use of standard tools. Greater depth-to-diameter ratios require special tools and can cause problems in controlling the diameter and form (shape) of the hole. Drilled and reamed holes are seldom perfect cylinders, but form variations are very small when holes are correctly produced. Design proportions of holes can influence the achievable form.

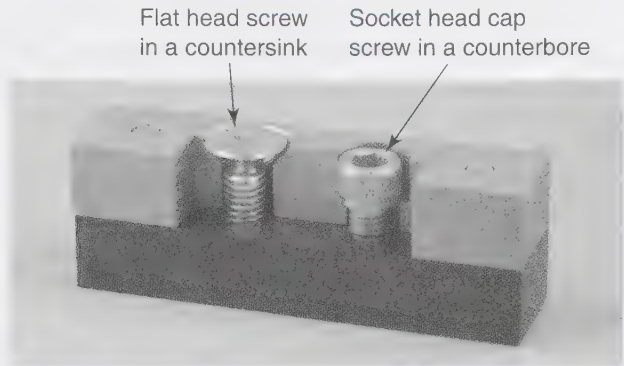


Goodheart-Willcox Publisher

Figure 4-9. Hole depth must be specified if the hole does not go through the part.

Holes for Machine Screws

Many functional applications require something more than a simple hole. As one example, flush screw heads require a counterbore or countersink. See **Figure 4-10**. Two different screws



Goodheart-Willcox Publisher

Figure 4-10. The screw heads in the shown parts are recessed in a countersink and counterbore.

are shown in the parts. The socket head cap screw requires a counterbore. The flat head screw requires a countersink to provide a bearing surface and also to permit the screw head to be flush.

Counterbores

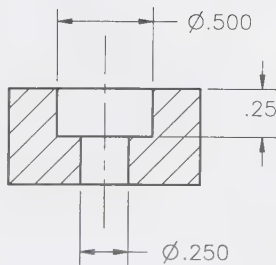
A **counterbore** is a stepped increase in the diameter of a hole. See **Figure 4-11**. The bottom of a counterbore is perpendicular to the axis. A counterbore, by design, is on the same axis as the hole. A small fillet may be included at the bottom of the counterbore. When a fillet is included, the counterbore cutting tool has a small radius ground on the corner, resulting in a fillet at the bottom of the counterbore. This fillet prevents a sharp corner that can cause stress concentrations, which can lead to cracking.

Counterbore holes are specified by giving the hole diameter, hole depth (when required), counterbore diameter, counterbore depth, and fillet radius. Counterbores may be dimensioned

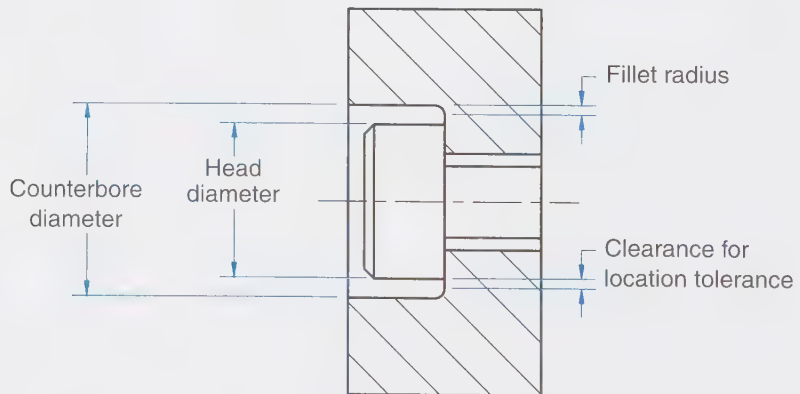


Depth specified by note

Remaining material dimensioned in section view



Section view dimensions



Bolt head clearance

Goodheart-Willcox Publisher

Figure 4-11. Counterbore dimensions provide a controlled depth.

by notation, applied dimensions, or a combination of the two. The counterbore symbol is used when calling out counterbore sizes. Counterbore is abbreviated CBORE when abbreviations are used in notes lists.

The distance from the surface to the counterbore bottom may be noted in the counterbore callout or dimensioned in a section view. It is also permissible to dimension the thickness of the remaining material to control the counterbore depth. The remaining material is dimensioned only when a critical need exists for a specific material thickness. Controlling and inspecting the remaining material thickness is normally more expensive than controlling the counterbore depth from the surface.

The counterbore note is attached to the hole by a leader. When using orthographic views, the leader is usually applied to the circular view. The leader points toward the hole's center and terminates with an arrowhead on the first object line.

A common use for counterbores is to recess fastener heads, such as those found on screws and bolts. To recess a feature such as a screw head, the counterbore diameter must be equal to the head diameter, plus two times the fillet radius, plus any location tolerance that affects the relative locations of the screw and counterbore. A counterbore diameter calculated in this way ensures that the head does not ride up on the counterbore fillet. To recess a bolt head, the minimum counterbore depth is equal to the maximum head height plus the minimum amount of recess desired.

Countersinks

A *countersink* is an angular increase in the size of a hole. It has the shape of a right circular cone and is coaxial with the hole. See Figure 4-12. Countersinks are used to recess fastener heads, such as those on flat head screws, and to break

(remove) the sharp edges of holes. Standard flat head screws have an included angle of 82° or 100°. The angle of the countersink must match the screw head angle. Countersink tools used to break sharp edges have a 90° included angle and are not meant to be used for recessing screw heads.

Countersunk holes are specified by giving the hole diameter, countersink diameter, and countersink angle. The included angle is always specified. The diameter of a countersink is the distance across the outermost diameter. Irregular surfaces make measurement of a countersink diameter difficult. See Figure 4-13. The countersink diameter on an irregular surface is equal to twice the minor radius of the countersink. The *minor radius* is the shortest distance between the axis of the hole and the intersection of the countersink with the surface. When a surface is uneven or curved, sizing the countersink to the minor radius will result in a screw head that is below some areas of the surface.

Countersink requirements may be given through notation, applied dimensions, or a combination of the two. The countersink notation is attached to the hole by a leader. The leader points toward the hole's center and terminates on the countersink.

Dimensions for a countersink are determined by the function of the countersink. A countersink used to recess a screw head has an angle equal to the angle of the screw head. The minimum diameter of a countersink must be equal to the sharp diameter of the screw head or the screw will protrude above the surface. See Figure 4-14. The diameter of the screw head before the sharp edge is removed is known as the *sharp diameter*. A countersink diameter equal to the sharp head diameter causes the screw head to be flush with the surface. A countersink diameter larger than the sharp head diameter will ensure that the screw head is below the surface.

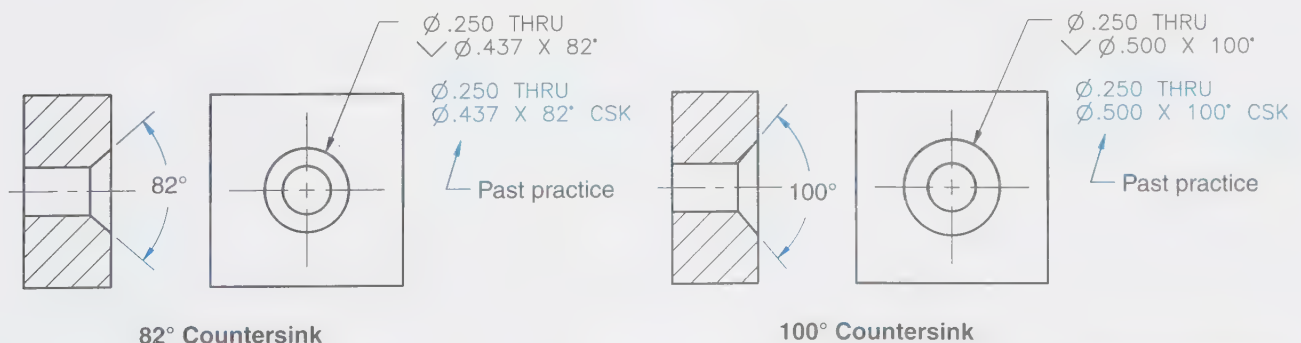
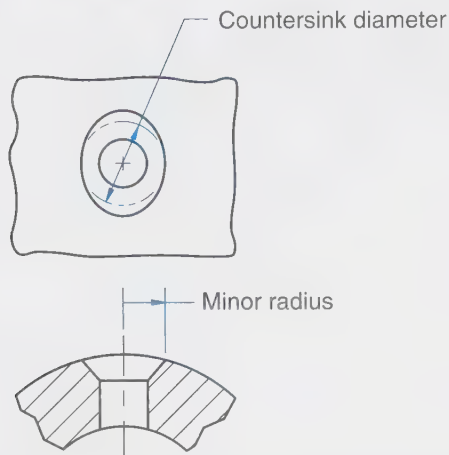
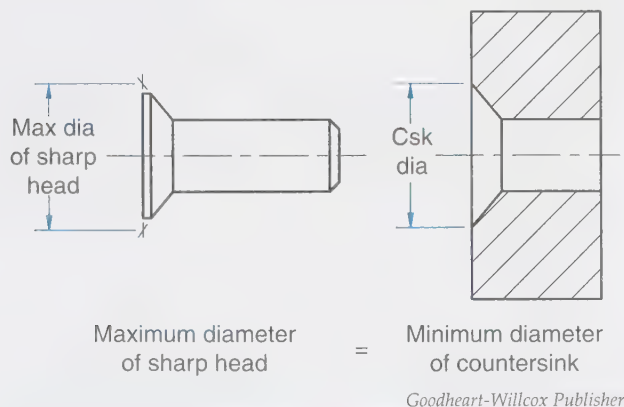


Figure 4-12. A countersink is used to recess a flat head machine screw or bolt head. A small countersink may also be used as a chamfer on a hole.



Goodheart-Willcox Publisher

Figure 4-13. The countersink diameter on an irregular surface is twice the minor radius.

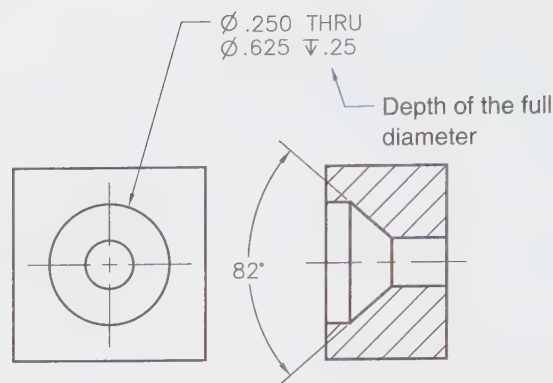


Goodheart-Willcox Publisher

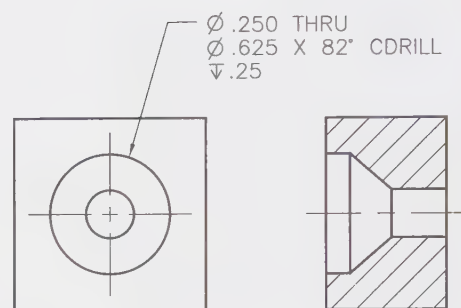
Figure 4-14. The countersink diameter should be at least the same size as the sharp diameter of the screw head.

Counterdrills

A counterdrilled hole is somewhat of a cross between a counterbore and a countersink. See [Figure 4-15](#). There is a stepped increase in the hole diameter. A conical transition in hole diameter is made by the counterdrill. The counterdrilled diameter is coaxial with the through hole.



Counterdrill angle dimensioned



Counterdrill angle included in notation

Goodheart-Willcox Publisher

Figure 4-15. A counterdrill creates a recessed countersink.

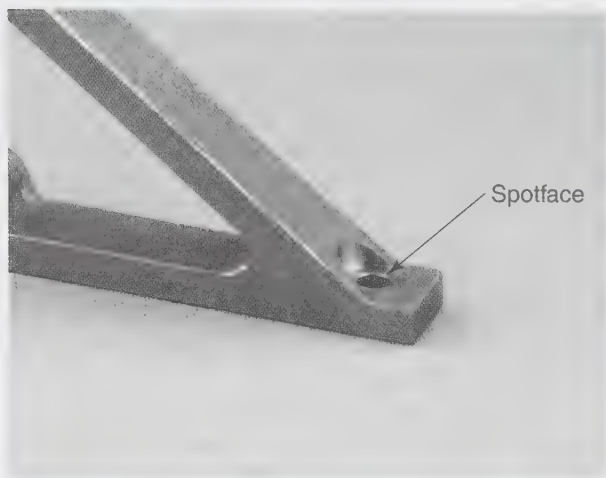
There is no symbol for a counterdrilled hole. Counterdrilled holes are specified by giving the hole diameter, counterdrill diameter, counterdrill depth, and, optionally, the counterdrill angle. It is recommended that the angle be specified when the remaining material or clearance is of importance. The counterdrill depth is the depth of the full diameter. The ASME Y14.5 standard only shows the counterdrill angle dimension applied in a section view, as shown in the upper segment of the given figure. However, industry practice generally uses notation of the angle instead of an angle dimension in a section view. The lower segment of the figure shows the noted angle with an abbreviation of CDRILL for counterdrill. Although commonly done in industry, showing the counterdrill angle in a notation may be seen as a noncompliance with the standard. The abbreviation is in compliance with ASME Y14.38.

The counterdrill specification is attached to the counterdrilled hole by a leader. The leader extends toward the center of the hole and terminates on the first object line.

Spotfaces

A *spotface* is similar to a counterbore, but does not go as far into the part. See [Figure 4-16](#). A spotface is commonly used to create a flat surface against which a screw head or washer can rest. It is not intended to recess the screw head.

The minimum acceptable amount of information for a spotfaced hole is the hole diameter and the spotface diameter. See [Figure 4-17](#). If no depth is given for a spotface, the spotface is made only to a depth sufficient to create a flat surface of the



Goodheart-Willcox Publisher

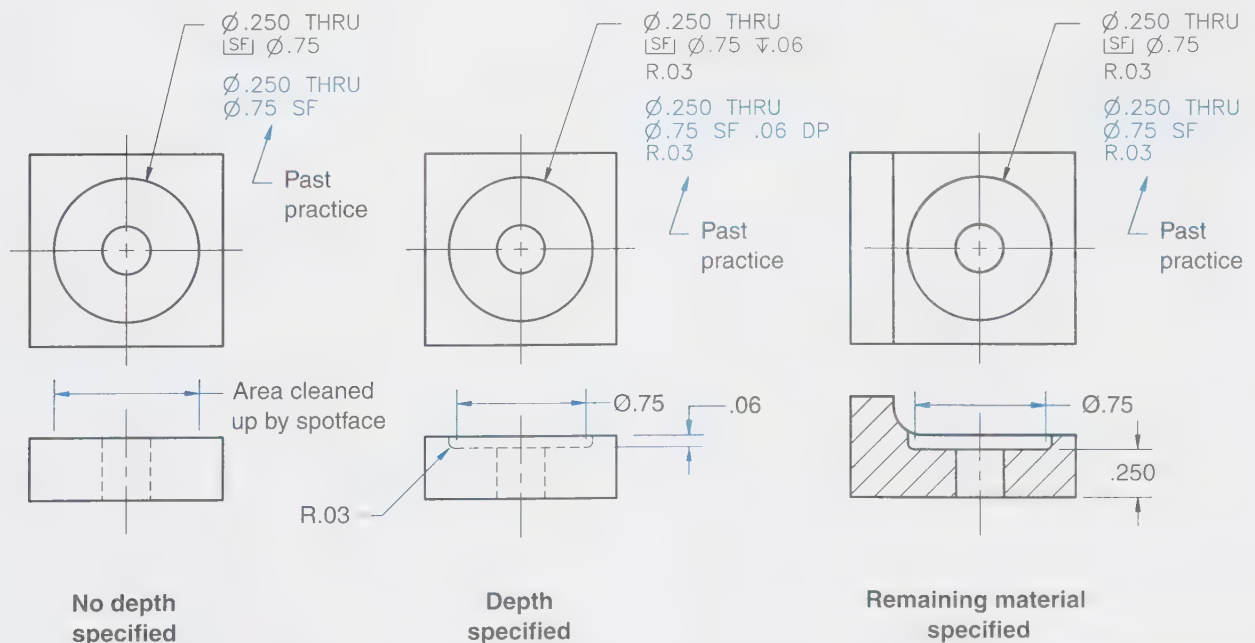
Figure 4-16. A spotface is used to clean up the surface on the cast part.

specified diameter. This method of dimensioning a spotface is not the best option for parts that are machined by numerical control.

The diameter specified for a spotface is the size of the flat produced on the surface. The measured diameter of the spotface does not include any fillet in the corner. There are two methods for specifying the required depth of a spotface, and they are the same as for a counterbore. One method is to specify the depth relative to the surface that is spotfaced. This depth is established on the basis of the depth needed to ensure a flat surface with all tolerances considered. The other is to dimension the remaining material in a section view. When this method is used, the remaining thickness is determined based on several factors, including the required strength of the part, the grip length of the fastener, and the depth needed to clean up the surface. The fillet radius for a spotface may be specified in the callout, given in a general note, or dimensioned in a section view. It is usually easiest to show it in the callout.

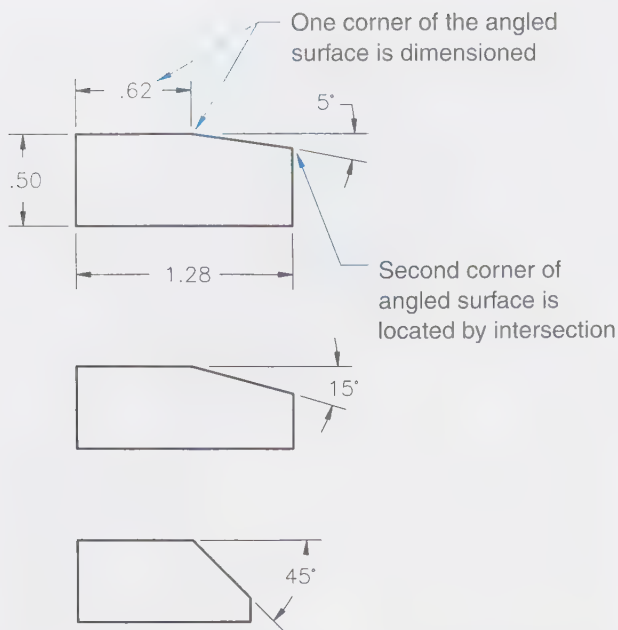
Angles

Angles may be dimensioned by showing the degrees between the features that are the sides of an angle. See [Figure 4-18](#). Angle values are specified by using one of two methods. One method uses degrees, arc minutes, and arc seconds; the



Goodheart-Willcox Publisher

Figure 4-17. A spotface is similar to a counterbore except that its diameter is the size of the flat created on the surface, and its specification may or may not include a controlled depth requirement.



Goodheart-Willcox Publisher

Figure 4-18. The arc used for a dimension line when dimensioning an angle is centered on the vertex of the angle.

other uses degrees and decimal parts of a degree. The symbols used in dimensioning angles are:

degrees	30°
minutes	30°15′
seconds	30°15′30″
decimal	30.25°

Tolerances on an angle are specified using the same units used for the nominal dimension value. The nominal angle dimension and tolerance have the same number of decimal places.

30.50° ±.25° (tolerance is <i>point two five</i>)
30°30′ ±0°15′
45°10′ ±1°0′

Surfaces or edges forming an angle are extended using extension lines, and a dimension line is drawn between the extension lines. The dimension line is an arc. The center point of the dimension line arc is located at the vertex of the angle. The extension lines, if extended toward the vertex, should intersect at the vertex of the dimensioned angle.

The dimension line arrowheads and dimension value may both be outside the extension lines, the arrowheads outside and the value inside, or the arrowheads and value inside. The arrangement of arrowheads and value depends on the angle size and the distance the dimension is located from the angle vertex.

One corner of an inclined surface must be dimensioned to define its location, and the angle of inclination specified. The second corner of the inclined surface is located by its intersection with another side of the part. If both ends of an inclined surface are located by coordinate dimensions, the angle does not need to be dimensioned.

Perpendicular surfaces in an orthographic view are understood to be at a 90° angle without any dimension being shown. All other angles must be dimensioned, or endpoints must be defined by coordinate dimensions.

Chamfers

A chamfer is a small inclined surface used to eliminate an abrupt edge on a part. It may be used to eliminate sharp edges and to facilitate assembly of close-fitting parts. See [Figure 4-19](#). Chamfers of 45° may be dimensioned by note. All other chamfer angles must be dimensioned in one of two ways. One way is to show the angle of inclination and the length of one side of the chamfer. The other way is to show the length of both sides of the chamfer. See [Figure 4-19](#).

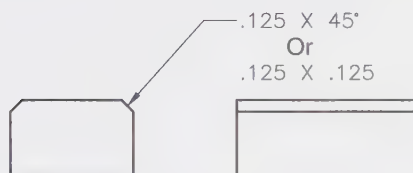
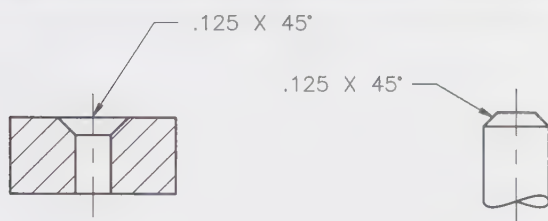
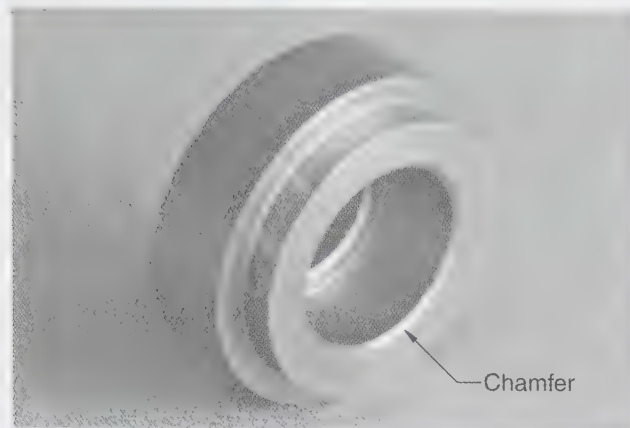
A note may be used for 45° chamfers because both sides of a 45° triangle are equal in length. The sides of any other angle are unequal and therefore cannot be specified by a note. A note cannot indicate which side of the angle is being given. The only way to be certain of control over chamfers other than 45° is to apply the chamfer dimensions to the part.

Tapers

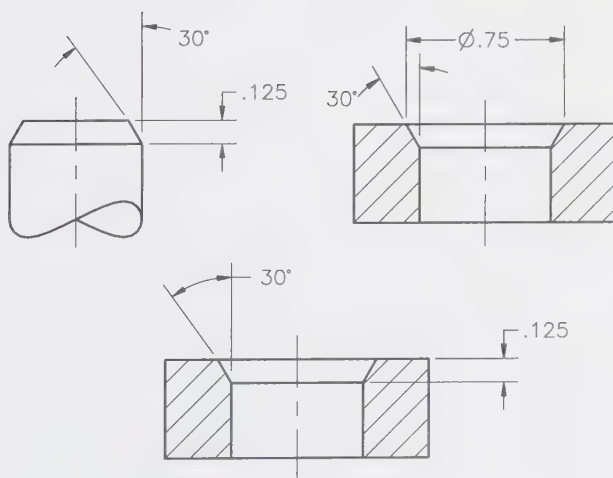
Tapers on a shaft or hole create a conical surface. Two purposes of tapers are to locate parts relative to one another and to hold parts together. A tapered pin sliding into a tapered hole provides a well-controlled location. The axes of two tapered parts coincide when the parts are assembled. A taper of the correct angle provides a good clamping force between assembled parts. Some standard tapers provide enough clamping force to hold cutters in the spindles of milling machines.

Several methods are acceptable for dimensioning tapers. See [Figure 4-20](#). The method used depends on the amount of control desired. Tapers on parts meant to mate existing machines and tools must be dimensioned using the appropriate standard machine taper. The machine taper to be mated can be determined by looking in the manufacturer's handbook that is supplied with any machine. Machine tapers are described in ANSI B5.10.

A standard machine taper may be dimensioned by noting the taper name and number. The diameter at one end and the length of the taper must be dimensioned. In the given example, the small end of the taper (.572") and the taper length (2.562") are dimensioned. American Standard Taper Number 2 is specified. The shaft made to these dimensions will mate a machine spindle with an American Standard Taper Number 2.



Chamfer size noted

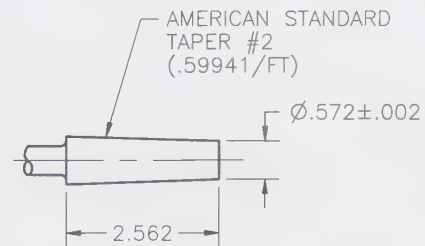


Chamfer size dimensioned

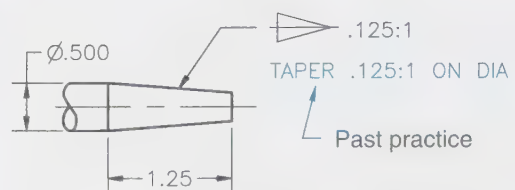
Goodheart-Willcox Publisher

Figure 4-19. Chamfers eliminate sharp edges and facilitate assembly of close-fitting parts.

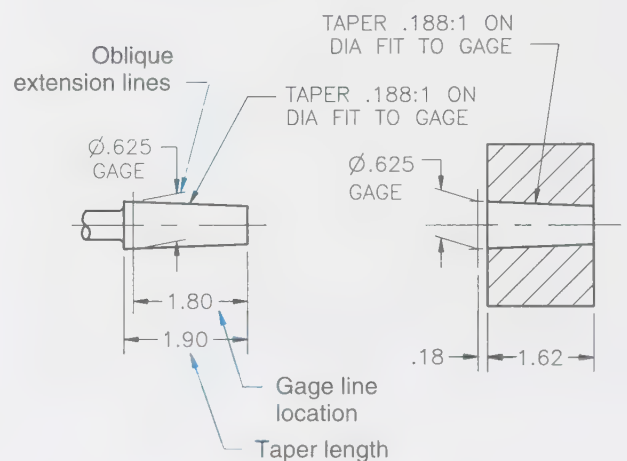
must be dimensioned. In the given example, the small end of the taper (.572") and the taper length (2.562") are dimensioned. American Standard Taper Number 2 is specified. The shaft made to these dimensions will mate a machine spindle with an American Standard Taper Number 2.



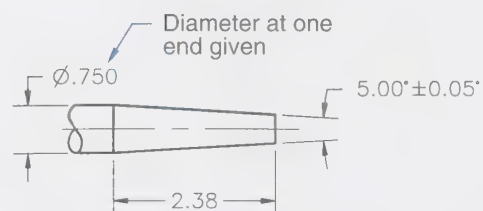
Standard taper specified



Taper per inch specified



Taper to gage specified



Included angle specified

Goodheart-Willcox Publisher

Figure 4-20. Tapers may be dimensioned through several methods. The application determines which method is appropriate.

Tapers that are not made to the standard sizes may be dimensioned by giving the taper on diameter per unit of length. Taper on diameter means the change in diameter. The diameter of one end, the length of taper, and taper on diameter per unit of length are dimensioned. The given example shows:

TAPER 125:1

The diameter of the tapered part must change .125" per inch of length.

Close control of nonstandard tapers may be dimensioned by specifying that the taper diameter be fitted to a gage. A position on the taper is dimensioned as the gage line location. A taper gage of the given diameter on one end will align with the gage location. Any error in the taper angle or diameter causes a substantial misalignment between the gage and gage line. Because some tolerance must be allowed on the taper, a relatively large tolerance is applied to the gage line location. The given example does not show a tolerance on the 1.80" dimension, and a large title block tolerance is assumed to apply. In addition to the gage line location and diameter, the taper on diameter per unit of length must be specified. The length of taper must also be controlled.

Noncritical tapers may be dimensioned by giving the included angle, length of taper, and diameter at one end. It is also permissible to omit the angle if the diameter at both ends is specified. It should be noted that determining the diameter of a tapered hole at a surface may be difficult if the edge of the hole is rounded or chamfered. Even if not rounded or chamfered, it may be difficult to precisely measure the diameter at the surface.

Arcs

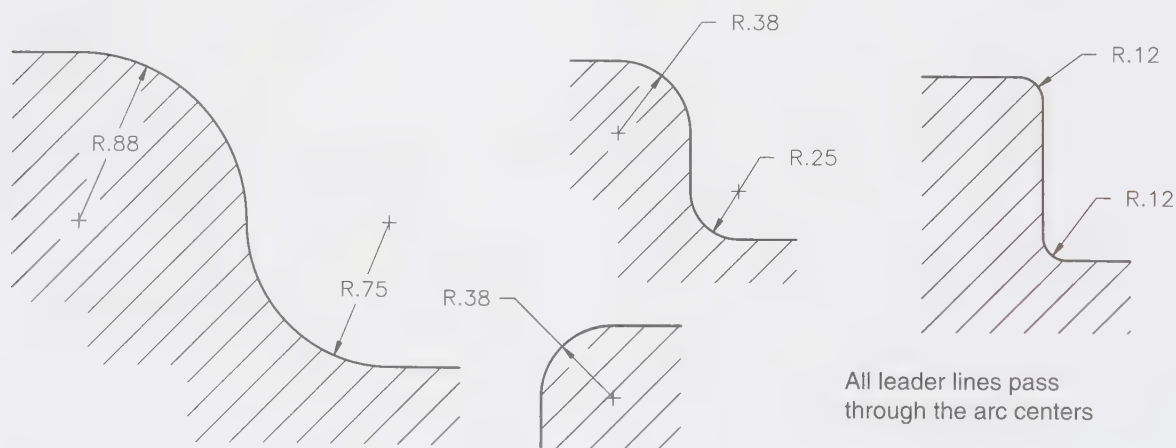
An arc is dimensioned by showing its radius. See [Figure 4-21](#). A leader line is oriented to pass through the arc's center. The arc center is shown by two short crossed lines.

When space permits, the leader line extends from the arc center to the arc. An arrowhead is placed on the end of the line that touches the arc. An arrowhead is never placed at the arc center. The dimension value is placed in a break in the leader line. The letter R is placed in front of the dimension value to indicate that it is a radius. Prior to the 1982 standard, the letter R followed the dimension value.

An arc with a small radius is dimensioned with the leader line extended to the dimension value. If the arc center is inside the object, a leader line runs from the arc center to the outside of the object. If the arc center is outside the object, the leader line extends from the arc, through the center, and to the dimension value.

An arc that is too small for the above practices is dimensioned with the leader line outside the object, regardless of whether the arc center is inside or outside the object. The leader lines are always oriented to pass through the arc center. The arc centers for small radius arcs are not shown. Showing the arc center for small radii would interfere with the dimension line or the dimension arrowhead.

Arc locations are defined in one of two ways; either the arc center is dimensioned, or it is located by *arc tangents*. See [Figure 4-22](#). Two short crossed lines are used to identify the arc center when the center is located by dimensions. The crossed lines are not required when an arc is located by tangents.



Arc dimensions

All leader lines pass through the arc centers

Goodheart-Willcox Publisher

Figure 4-21. Arcs are always dimensioned by giving the radius.

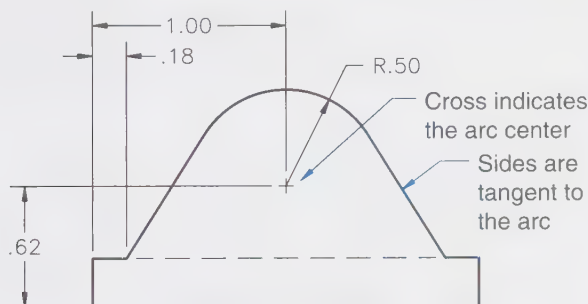
The positions of tangents are determined by the arc's location if the arc center point is dimensioned. If the arc center point is not dimensioned, then the tangent locations must be dimensioned. When tangents are dimensioned, the arc is located by the tangents. It is incorrect to provide dimensions for both the arc center and the arc tangents. Providing both is double dimensioning and is not acceptable.

The center for an arc with a large radius can be difficult to show in its true position. See **Figure 4-23**. In orthographic views where the true position of the center would cause interference with other features or views, or fall off the drawing sheet, the arc center may be shown out of position for dimensioning purposes.

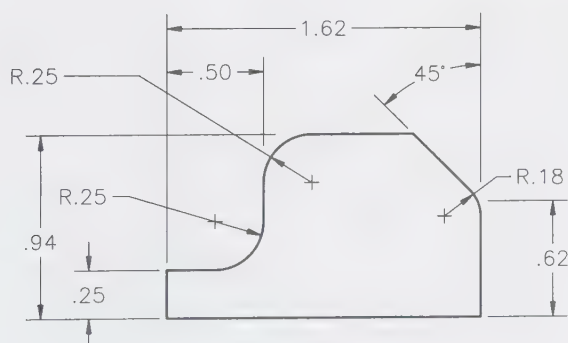
The true arc center position must be used for drawing the arc. A simulated arc center is then shown in a position that is convenient for dimensioning. The position is selected to minimize the resulting offset in the radius dimension line.

The dimension line is drawn in segments with an offset included. The segment of the dimension line that touches the arc is radial to the arc.

Dimension lines locating the arc center are shortened. Although dimension lines are shortened,



Arc center located by dimensions



Arcs located by tangents

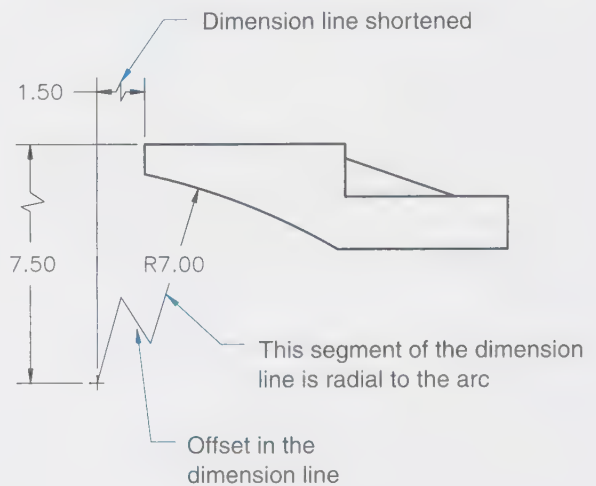
Goodheart-Willcox Publisher

the shown dimension values are the true location dimensions for the arc center. This practice is not used with CAD models.

Foreshortened Radii

The radius of an arc is dimensioned in a true shape view whenever a true shape view is given. If no true shape view exists, the true radius may be dimensioned wherever a foreshortened view of the arc is seen. See **Figure 4-24**. A foreshortened view is seen when the line of sight is inclined to the surface on which an arc exists. Foreshortened views and dimensions are not applicable to CAD models.

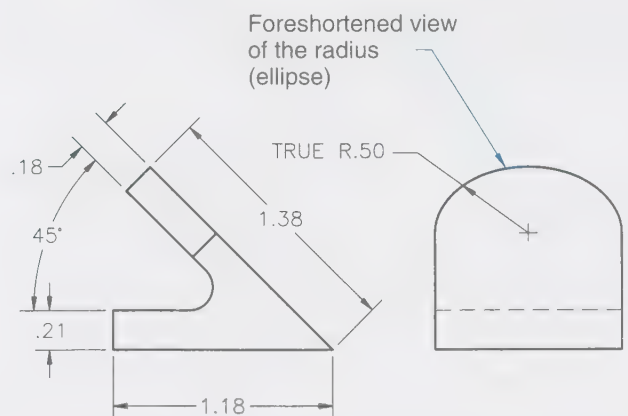
The dimension line for a foreshortened radius extends from the arc center to the arc. An arrowhead



Large radius arc

Goodheart-Willcox Publisher

Figure 4-23. The true dimensions defining arc center location must be shown even when the center point is not shown in its true location.



Goodheart-Willcox Publisher

Figure 4-24. The true radius of an arc must be specified even if the dimension is shown on an elliptical view of the arc.

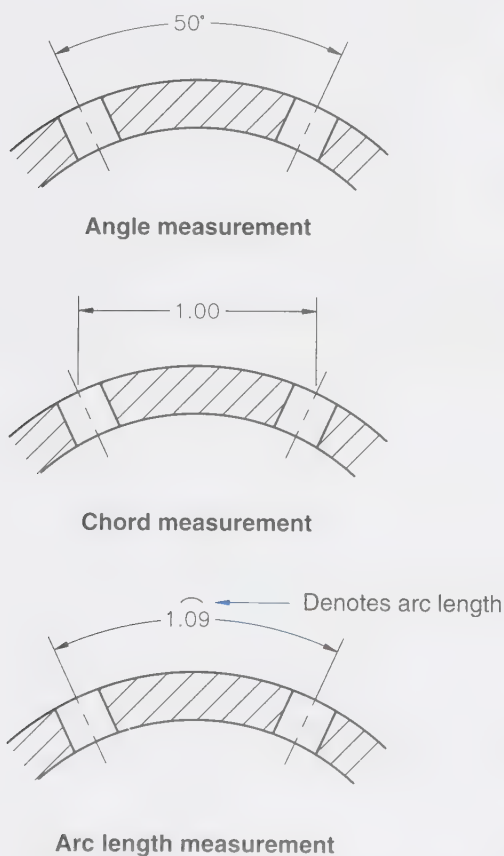
is placed at the arc. The dimension value may be placed either inside or outside the object, depending on the size of the radius and available space. A prefix of TRUE R is placed on the radius dimension.

Features on Curved Surfaces

Features on a curved surface are located by one of three methods. See **Figure 4-25**. The angle, chord, or arc length may be given.

Location by a chord is an acceptable method of location, but fabrication accuracy can be difficult to check manually. This is especially true if the located features are holes normal to the curved surface. The reason for difficulty in checking chord measurements between radial holes is that the chord is specified in relationship to the curved surface. This reference surface is cut away as the holes are drilled, making it necessary to mathematically determine where the surface would be based on the surface area that remains.

An arc length dimension is an acceptable method for locating features on a curved surface, but it too can be difficult to check. Arc length is the distance measured on a curved surface. The dimension line used for an arc length dimension is



Goodheart-Willcox Publisher

Figure 4-25. The method used to locate features on a curved surface depends on the function of the part.

an arc and has the same center as the dimensioned feature. A small arc is placed above the dimension value to indicate that it is an arc length. This value can be difficult to measure for the same reasons that pertain when the chord measurement is specified.

Spherical Radii

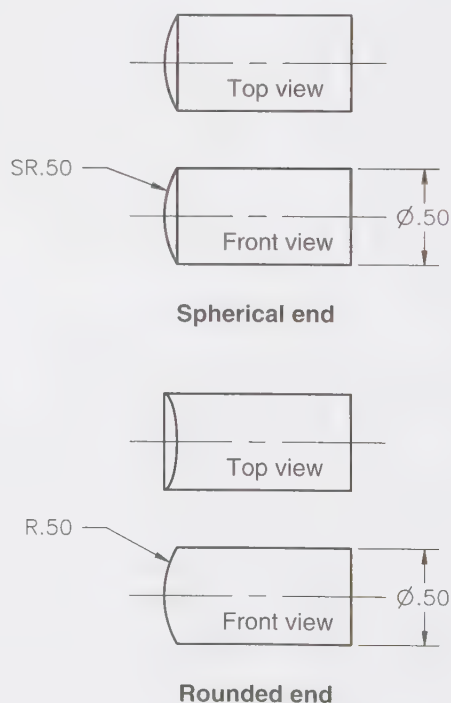
A spherical radius must be specified by placing the letters SR in front of the dimension. See **Figure 4-26**. Failure to show the SR prefix could result in a simple arc being produced on the part. A spherical radius is easiest to produce and verify when the location dimension goes to the outside of the spherical surface.

Prior to the 1982 standard, it was required that the suffix SPHER R be applied to spherical radii dimensions.

Irregular Curves

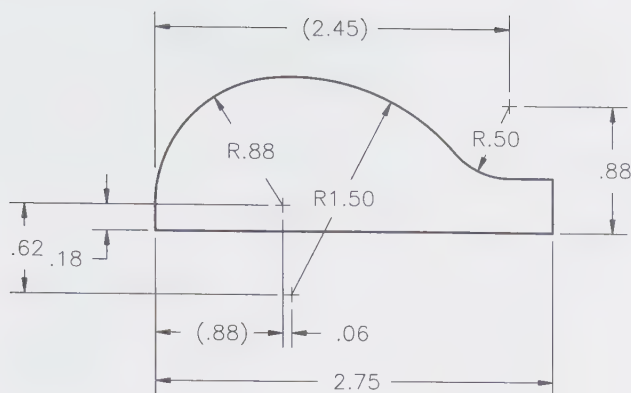
Curved surfaces made of tangent arcs are dimensioned by showing the locations and sizes for the arcs. See **Figure 4-27**. In the given example, the position of each arc is determined by a combination of arc center location dimensions and tangent feature locations. Each radius is dimensioned.

When the curve is not a common geometric shape, the curve is dimensioned with coordinates locating points along the curve or a mathematical formula defining the curve is given. When

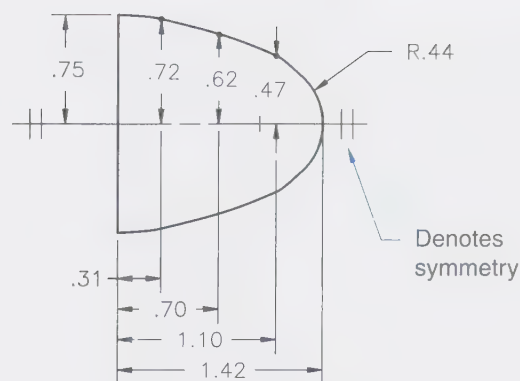


Goodheart-Willcox Publisher

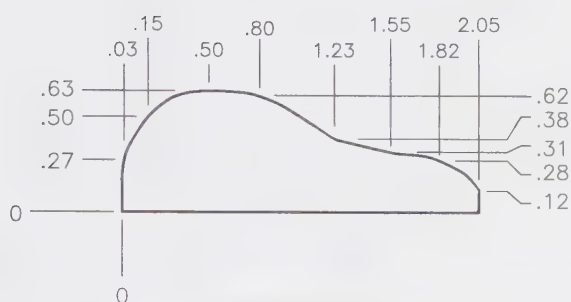
Figure 4-26. A spherical radius is noted by adding the spherical radius prefix to the dimension.



Tangent arcs



Symmetrical curve



**Irregular curve
(continuously changing radii)**

Goodheart-Willcox Publisher

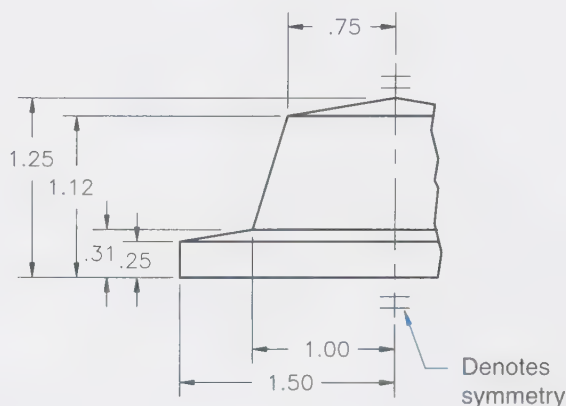
Figure 4-27. Curved surfaces are sometimes dimensioned by showing point locations along the curve.

dimensioning points on a curve, a sufficient number of points must be dimensioned to ensure production of a smooth curve. It is also possible to define the requirements mathematically or with a CAD-defined surface. When curved surface dimensional accuracy is required, a mathematical or CAD definition of the surface is recommended.

Symmetrical Features

Only half of a symmetrical part must be dimensioned when using orthographic views. See [Figure 4-28](#). The dimensions shown must define the location and size of each feature relative to the line of symmetry. All locations and sizes are shown relative to the line of symmetry and are known to repeat on the opposite side of the line. The dimensions are known to repeat because symmetry is indicated on the drawing by two short lines drawn across each end of the centerline.

A symmetrical curved part is dimensioned in [Figure 4-28](#). The centerline is indicated to be a line of symmetry. The distance from the centerline to the curve is dimensioned in four places. The locations for each of the four dimensions are given.



**Partial view of a
symmetrical part**

Goodheart-Willcox Publisher

Figure 4-28. The dimensions for a symmetrical part are identical on both sides of the line of symmetry.

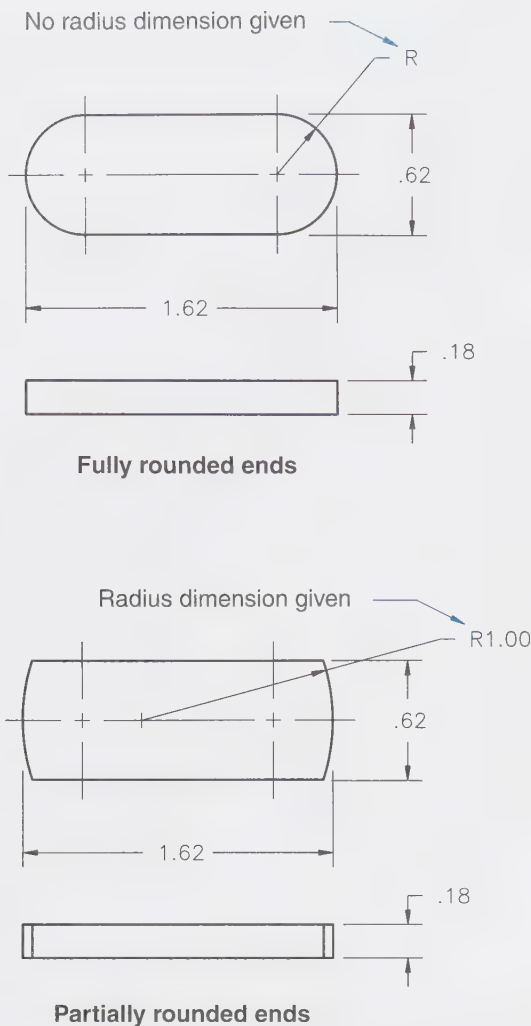
The undimensioned side of the part is identical to the dimensioned side because a line of symmetry is identified.

Large symmetrical parts may be shown in partial or half views. The given view shows slightly more than half the part. A break in the part is made just past the line of symmetry. Dimensions are shown relative to the line of symmetry when a partial view is used.

Round-Ended Bars

Round-ended bars come in two general shapes: fully rounded ends and partially rounded ends. Fully rounded ends have a radius that is tangent to the two sides. The radius of a fully rounded end is half the distance between the two tangent sides. The radius for partially rounded ends is always greater than half the distance between the two sides. A radius less than half the distance could not intersect both sides.

Examples of how to dimension round-ended bars are shown in [Figure 4-29](#). The overall length,



Goodheart-Willcox Publisher

Figure 4-29. The overall length of round-ended bars is dimensioned.

thickness, and the distance between sides are given. A radius value is not normally specified for fully rounded bars. The radius dimension line and the letter R are shown without giving a size value. No size is shown for the radius because it must be equal to half the distance between tangent sides. It is incorrect to show both a radius dimension and the distance between tangents on a fully rounded bar because showing both is double dimensioning.

If the radius on a fully rounded end is critical, then the radius may be dimensioned and the distance between sides omitted.

The radius for partially rounded ends must always be specified. The distance between sides has no effect on the radius size except to determine the minimum possible radius.

Slotted Holes

Slotted holes can be used when tolerances accumulate between parts to potentially prevent

fasteners from passing through round holes. Slotted holes can also be used to allow adjustment in the position of a part.

Three common dimensioning methods exist for dimensioning slotted hole size. See [Figure 4-30](#). The method used depends primarily on the design function but may also be influenced by the machine process that will be used to produce the slot.

Slotted holes punched in sheet metal are normally dimensioned by giving the overall length and width. An R is shown on one end to indicate that both ends of the slot are made on a radius that is tangent to both sides of the slot. These dimensions may be applied directly to the slot or shown in a notation that is attached with a leader.

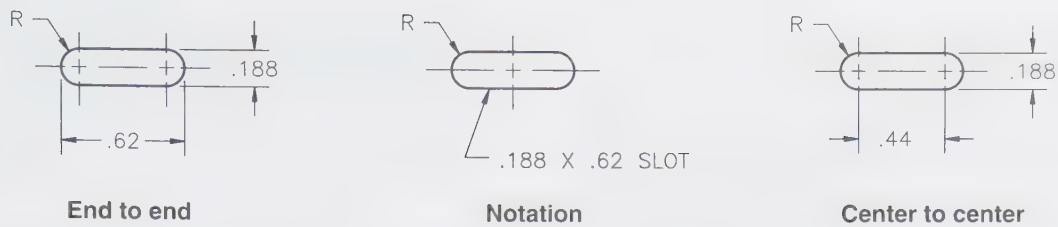
Slotted holes produced by machine cutting processes are commonly dimensioned by showing the slot width and distance between arc centers. The slot width is equal to the cutter diameter, and the center distance equals the amount of cutter travel. Dimensioning in this manner does not require the slot to be fabricated as explained. The slots may be cut using other methods provided the dimensional requirements are met.

One method used to dimension slot position is to define the center location. See [Figure 4-31](#). This method works well in combination with position tolerances. Other location methods may be used and the given figure shows two common alternate practices.

Keyseats

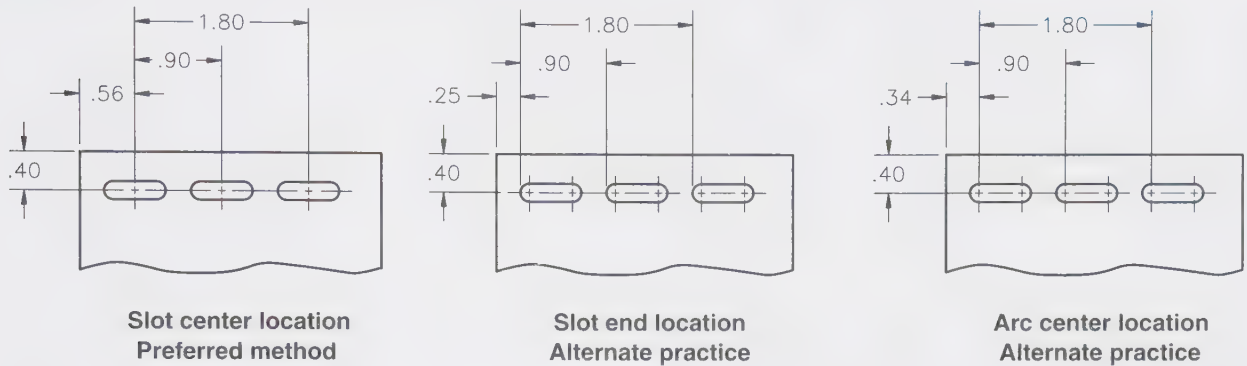
A *keyseat* is a recess cut into a shaft or hub. A key inserted into the keyseats of mating parts prevents a shaft from spinning inside the hub. Keyseat size is closely controlled to ensure a good fit between the keyseat and key. Keyseat depth and width must be dimensioned.

Depth of a keyseat is typically dimensioned from the far side of the shaft or hole. See [Figure 4-32](#). Dimensioning depth in this manner provides an existing feature from which dimensions can be checked. It also ensures that a key inserted in the keyseat will have a known height from the opposite side of the shaft or hole. The width is dimensioned as shown. The keyseat is not assumed to be perfectly centered on the shaft. A position tolerance must be applied as defined in following chapters to define the allowable amount of location error. Keyseat depth may be dimensioned from the center of the shaft or hole using basic dimensions when the center serves as a datum and profile tolerances are applied to the keyseat surface. Datums and profile tolerances are explained in later chapters.



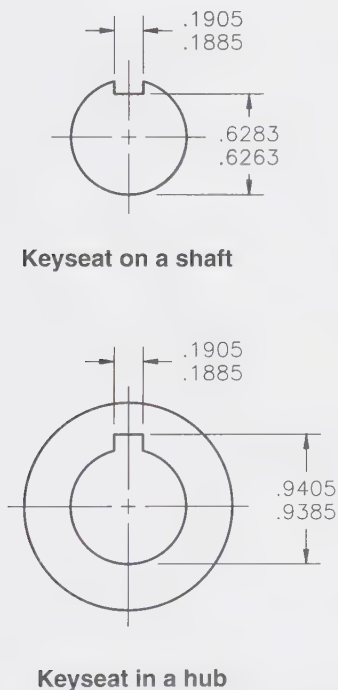
Goodheart-Willcox Publisher

Figure 4-30. These are the three methods of dimensioning the size of a slot.



Goodheart-Willcox Publisher

Figure 4-31. Depending on how the slot is created and applied, select one of these methods for dimensioning its location.



Goodheart-Willcox Publisher

Figure 4-32. The depth of a keyseat is always dimensioned from the far side of the shaft or hole.

Narrow Spaces

Small adjacent features may require several dimensions in a small amount of space. These dimensions are located at different distances from the part in order to fit in the narrow space and

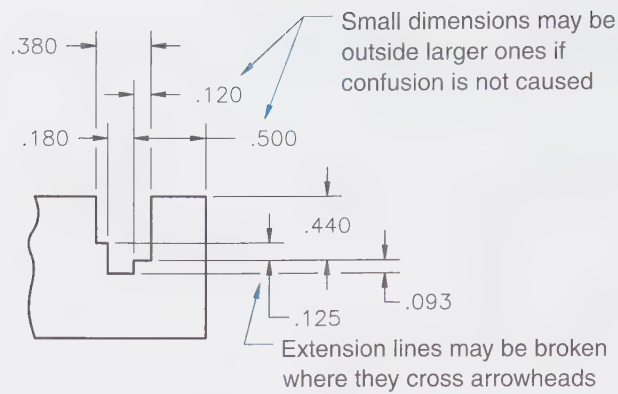
maintain legibility. See [Figure 4-33](#). The shown extension lines are relatively close together and may be broken where they cross arrowheads. This practice is not typically used in CAD models. The breaks help to clarify the point of application for each arrowhead. A combination of arrowhead and dimension value placements is used to maximize the clarity of dimension application.

A *joggle* (offset) is drawn in any extension line that is parallel within .06" of another line. This practice is not typically used in CAD models. The joggle is used to offset the extension line sufficiently to provide at least the .06" distance. See [Figure 4-33](#). A joggle allows clearance between adjacent lines without disconnecting the extension line from the point of application. A joggle in an extension line is preferable to exaggerating the size of a feature.

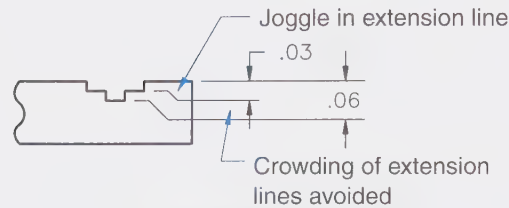
When joggles do not result in a clear application of dimensions, it is necessary to draw a larger-scale view of the dimensioned features. See [Figure 4-34](#).

Centerdrilled Holes

A centerdrilled hole is a hole and countersink cut into the end of a shaft or other cylindrical feature. One use for the centerdrilled hole is to mount the part in a lathe. A machine center on a lathe is inserted into the centerdrilled hole to ensure that the shaft turns on the lathe's axis of rotation when it is being turned.



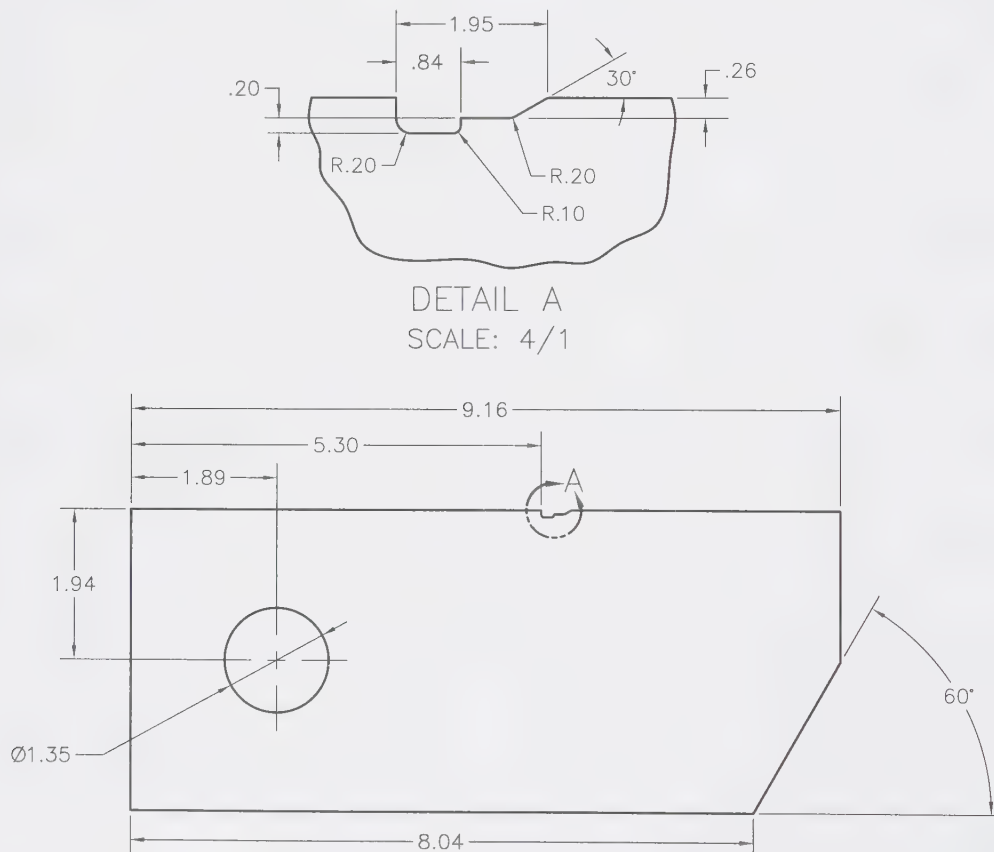
Offset dimensions



Offset extension lines

Goodheart-Willcox Publisher

Figure 4-33. Smaller dimensions are normally inside larger dimensions, but this can change to reduce breaks in extension lines and improve readability. The joggles of offset extension lines prevent the overlap of dimensions when features are located close to each other.



Goodheart-Willcox Publisher

Figure 4-34. When joggles are not enough to improve the clarity of a dimensioned feature, create a larger-scale detail view.

A special tool is used for centerdrilling processes. The tool is sometimes identified as a combined drill and countersink, but is commonly known in the trade as a *centerdrill*. Common sizes for centerdrills are standardized. All standard centerdrills have a 60° angle countersink.

The centerdrill size is noted on a drawing by specifying the number of the centerdrill to be used. See [Figure 4-35](#). The depth of the center-drilled hole is controlled by the maximum diameter of the countersink.

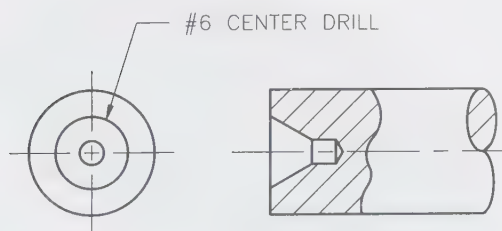
Machining Reliefs

A machine relief provides an area in which cutting tool travel can be stopped without damaging the tool or part. See [Figure 4-36](#). As an example, a relief permits a single-point threading tool to emerge from the machined part and run freely. As another example, a grinding wheel can travel sufficiently past a properly located relief to ensure the cut surface is machined evenly and without fear of making contact between the grinding wheel and the shoulder. Side pressure on a grinding wheel can cause it to disintegrate, so relief is necessary.

Machine reliefs are specified in a note or dimensioned in a detail view. In either case, the relief width, depth, and corner radius are given. Dimensions for the relief are determined by the space needed to clear the tools to be used during fabrication.

Threads

The minimum amount of information that must be given for a thread is the nominal size, threads per inch, thread form, and thread class. See [Figure 4-37](#). If the thread does not go all the way through the part, the thread depth must also be given. Thread depth is the distance that fully formed threads extend into the part or feature.



Angle and depth dimension not required

Goodheart-Willcox Publisher

Figure 4-35. A machine center hole is called out by specifying the centerdrill tool number.

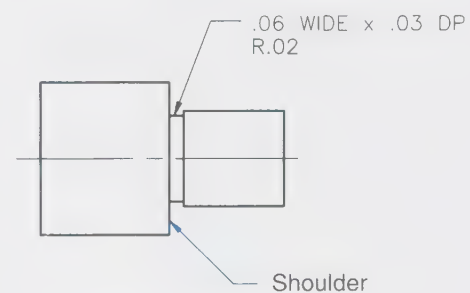
Knurls

Knurls are machined on cylindrical shapes to increase the friction between parts that are pressed together, improve the grip of a handle, or improve the appearance of parts. The purpose of the knurl is determines the type of dimensions given. See [Figure 4-38](#).

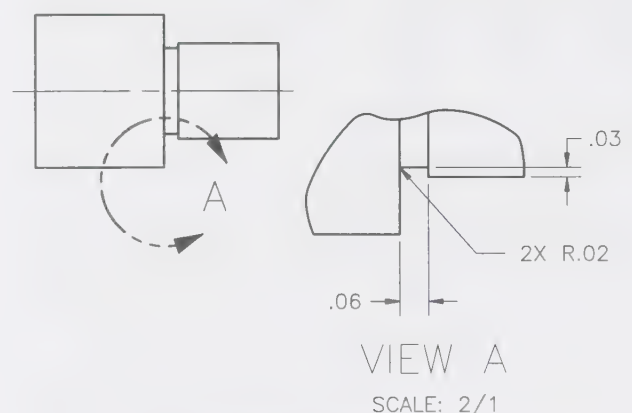
The diameter before and after knurling must be given on a shaft that fits into another part. The before and after knurling dimensions must include tolerances that show the acceptable amount of variation. Length of the full knurl, the diametral pitch (DP), and knurl form are all specified. Knurls used for appearance do not require specification of the after-knurling diameter.

Sheet Metal Bends

Two intersection lines (sometimes called mold lines) are created when sheet metal is bent. See [Figure 4-39](#). Intersection lines are located by extending flat surfaces that lie on the same side of the sheet metal. Extending the surfaces inside a bend creates



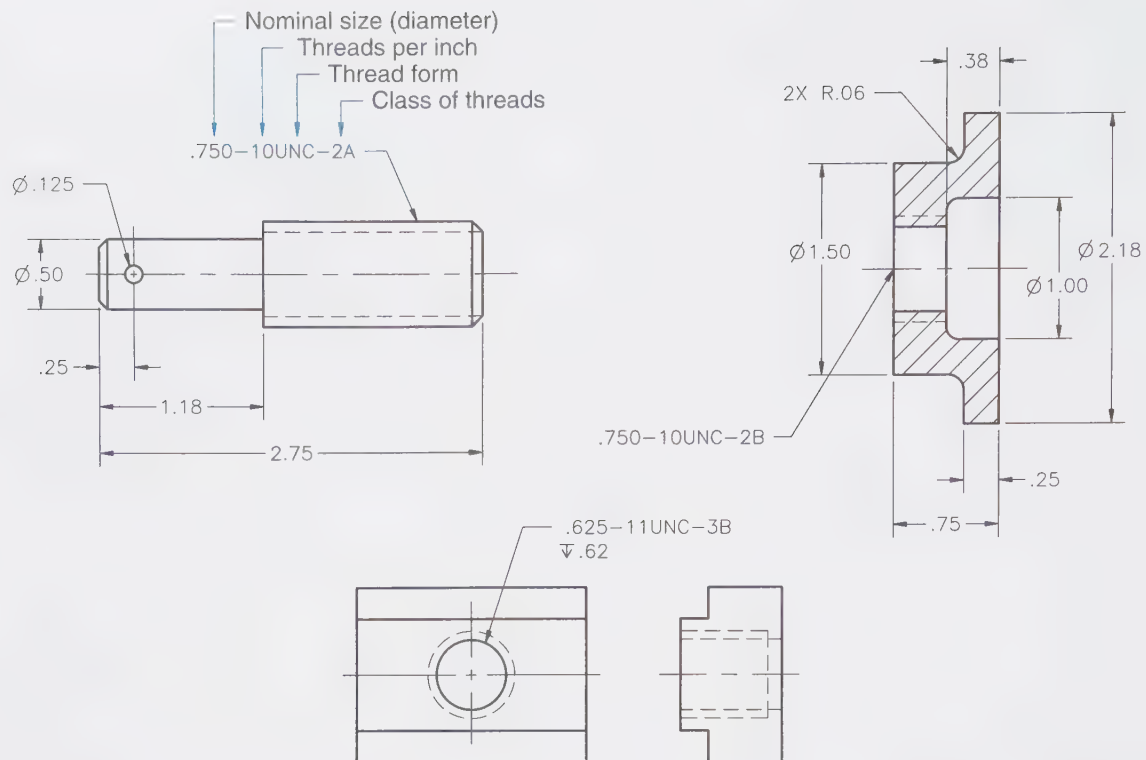
Noted relief dimension



Detail relief dimensions

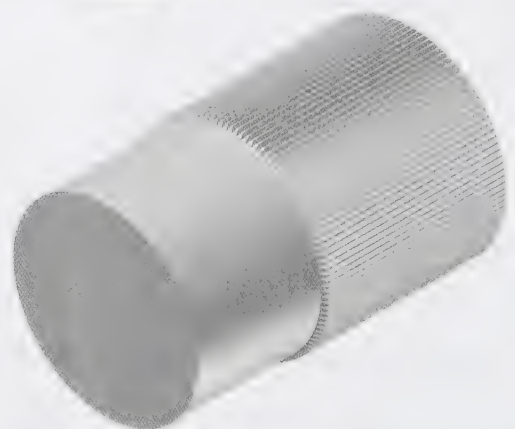
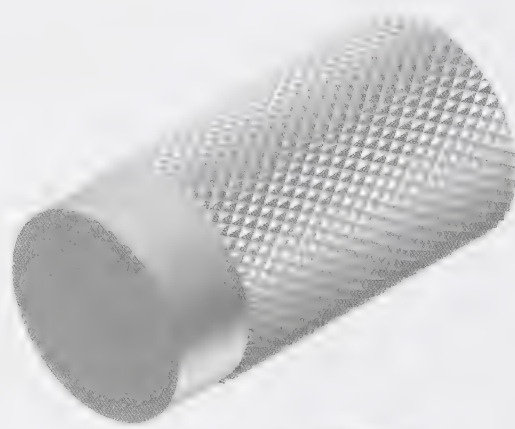
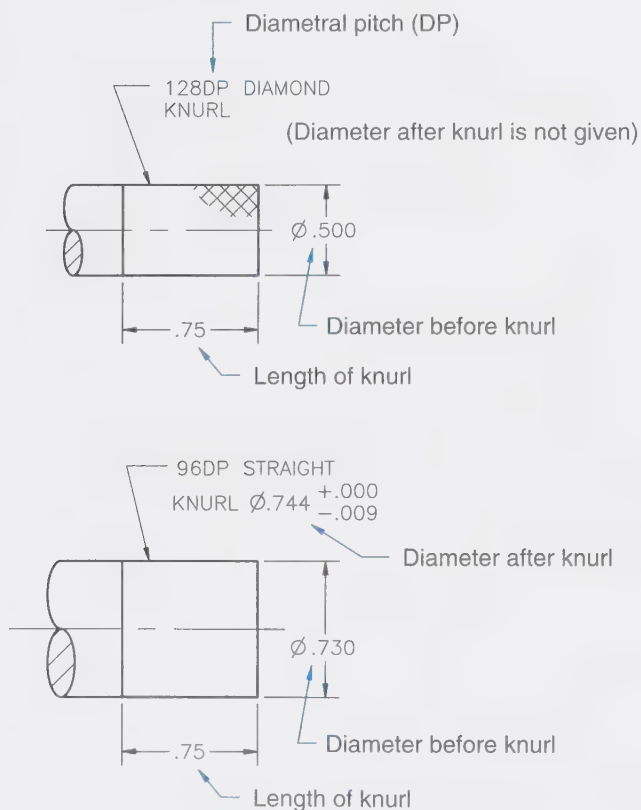
Goodheart-Willcox Publisher

Figure 4-36. Machine reliefs provide an area in which cutting tools are stopped without damaging finished surfaces.



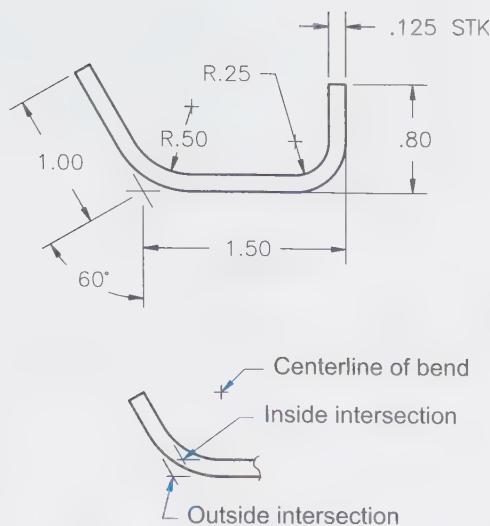
Goodheart-Willcox Publisher

Figure 4-37. The nominal size for Unified National Coarse threads is approximately equal to the maximum size limit of the major diameter on an external thread.



Goodheart-Willcox Publisher

Figure 4-38. Knurls can perform functional and aesthetic purposes.



Goodheart-Willcox Publisher

Figure 4-39. Sheet metal dimensions extend to intersections.

an inside intersection. Extending surfaces outside the bend creates an outside intersection.

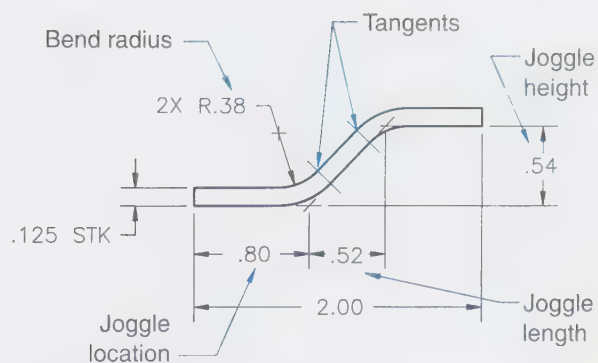
Sizes of sheet metal parts are defined by extending dimensions to intersection lines. The centerline of a bend is not normally dimensioned. The centerline of a bend is located by the tangent sides and the bend radius. Bend radii are specified by showing the inside radius. The bend radius used for a sheet metal part is primarily a function of the design application. However, a bend radius must not be smaller than can be tolerated by the material. A bend radius that is too small causes cracks in the material. The smallest allowable bend radius is a function of thickness and mechanical properties of the material.

Joggles

A *joggle* in a part is a transition between offset surfaces. Joggles on parts are often seen on sheet metal parts and are not to be confused with joggles in extension lines. See **Figure 4-40**. Joggles are dimensioned by giving the location, height, length, and bend radius. A straight (flat) segment between the bend tangents is recommended. This is especially true if the part is to be produced by a brake forming machine. A brake forming machine bends metal along a straight axis, and it is easier to make the bends if there is a straight (flat) segment adjacent to the bend.

Standard Sizes and Shapes

Production costs are kept down when standard-size parts and stock sizes can be used in designs. See **Figure 4-41**. The use of standard parts reduces the number of special operations



Goodheart-Willcox Publisher

Figure 4-40. A sheet metal joggle requires two bends. The tangents of the two bends must not overlap.

needed. Every special operation that is eliminated from the production of a design means increased profit for the company and reduced cost to the consumer. Standard sizes for screws, bolts, washers, pins, nuts, and other hardware already exist. Standard sizes for wires, sheet metal, metal plates, bar stock, and extrusions also exist. There is no need to dimension all the features on a standard part, because dimensions and tolerances are already specified in a standard part drawing.

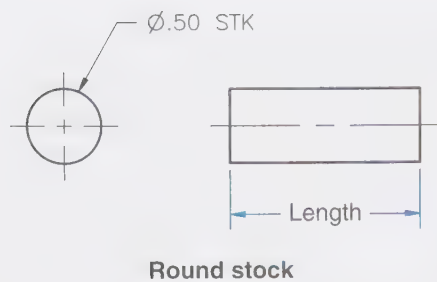
In addition to using national standards, individual companies often standardize parts and features that they use repeatedly. A panel cutout for an electrical connector is an example. The standard cutouts are documented in a company standards book and assigned an identifying number. The number is then referenced on drawings to avoid the need to repeat all the standard dimensions. One disadvantage of referencing a standard cutout is that a producing company may be different from the designing company, so the producing company would need to be provided a copy of the standard.

Broken Lengths

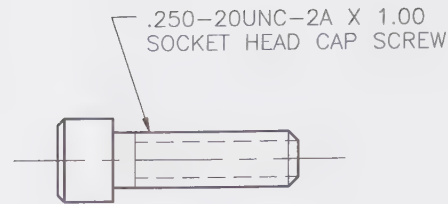
Long parts may be broken and a portion removed to reduce the drawing space required for the part. This practice is not used for CAD models. See **Figure 4-42**. The dimension used to show the length is not broken. The dimension value shown is the size to which the part is to be made.

Limits of Size

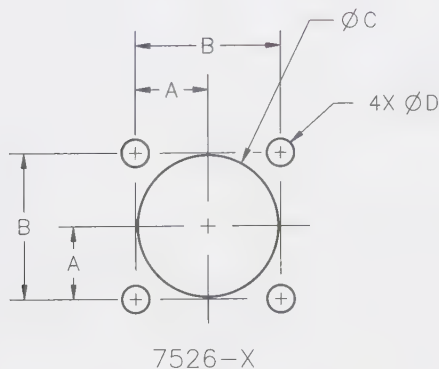
All dimensions, except basic dimensions, include a tolerance. *Tolerance* is the acceptable amount of variation that is allowed on a feature. A general tolerance shown in the title block or drawing notes is applicable to each dimension that does not show a specific tolerance applied to it.



Round stock



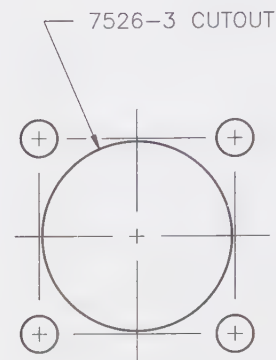
Fastener



7526-X

DASH NO (X)	A	B	C	D
-1	.50	1.00	.969	.188
-2	.62	1.25	1.188	.219
-3	.75	1.50	1.438	.250

Standards book specification

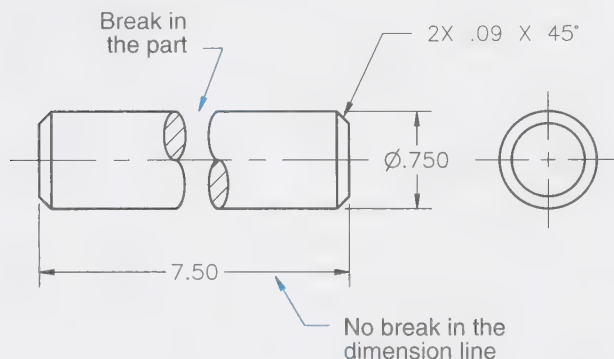


Drawing callout

Standard feature dimensions may be located in a company standards book

Goodheart-Willcox Publisher

Figure 4-41. It is not necessary to specify all the dimensions for standard stock shapes or for features specified in company standards books.



Goodheart-Willcox Publisher

Figure 4-42. Part of an object may be removed to shorten a view if the removed portion is a continuation of the shown features.

A tolerance applied on a size dimension defines the acceptable limits of size. There are several methods for applying tolerances to size dimensions. These include limit dimensions, bilateral

tolerances, unilateral tolerances, and single limit dimensions.

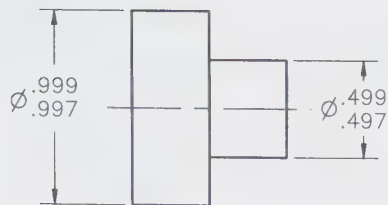
Limit Dimensions

Limit dimensions specify the minimum and maximum acceptable values. See [Figure 4-43](#). The difference between the minimum and maximum limits is known as the tolerance. The maximum limit is written above the minimum limit when both values are shown in the dimension. A line is not drawn between the two values.

If the limits are specified in a note, then the smaller limit precedes the larger value. The values are separated by a dash when shown in a note.

Plus and Minus Tolerances

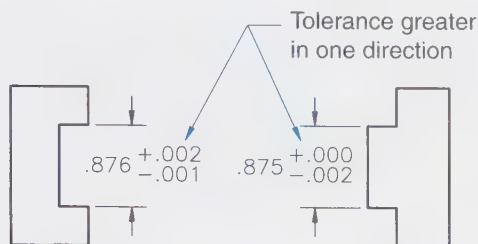
Plus and minus tolerances specify the acceptable amount of variation in both directions from the specified dimension value and may be either bilateral or unilateral. See [Figure 4-44](#). The amount



Limit dimensions

Goodheart-Willcox Publisher

Figure 4-43. The maximum and minimum acceptable size of a feature may be shown by using limit dimensions.



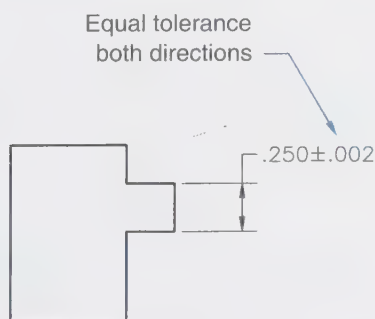
Goodheart-Willcox Publisher

Figure 4-44. The plus and minus tolerance values applied to a dimension may specify unequal amounts of allowed variation.

of acceptable variation is shown adjacent to the specified dimension.

Bilateral tolerances show an acceptable amount of tolerance in both a plus and minus direction. With a **unilateral tolerance**, either the plus or the minus value is zero. Even when the tolerance in one direction is zero, both tolerance values must be shown.

A bilateral tolerance may be shown with one tolerance value that permits an equal amount of variation above and below the specified dimension value. See **Figure 4-45**. The tolerance value is preceded by a plus-or-minus sign (\pm). If the dimension is a diameter, the diameter symbol is placed before the as-designed size.



Goodheart-Willcox Publisher

Figure 4-45. The plus and minus tolerance values applied to a dimension may specify equal amounts of allowed variation.

Single Limit Dimensions

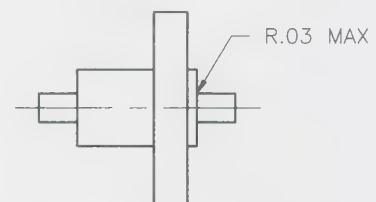
Single limit dimensions are acceptable only when the second limit is controlled by the part geometry. Calling out a R.03 MAX dimension indicates that a minimum radius of .00" is acceptable. See **Figure 4-46**. Calling out a minimum knurl length of .50" indicates that a maximum length knurl can be as long as the diameter on which the knurl is rolled.

Tolerances on Angles

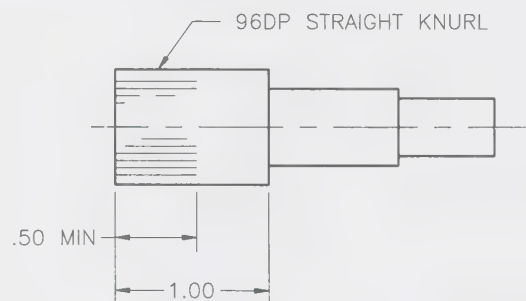
Tolerances on angles may be specified using limit dimensions or plus-and-minus tolerances. Angles may also be controlled with orientation tolerances as explained in a later chapter. Single limit dimensions are not practical for use on angles.

Calculation of Size Tolerances

Standard tables for different classes of tolerances may be used to determine the limits of size for mating features. Limits of size may also be calculated without the use of standard tables. Whether or not standard tables are used, the allowance and tolerances applied to the parts are based on the functions of the parts. *Allowance* means the



Maximum limit dimension



Minimum limit dimension

Single limit dimensions

Goodheart-Willcox Publisher

Figure 4-46. A single limit dimension establishes one acceptable size. The part geometry controls the other limit.

difference between the minimum hole size and the maximum shaft size and may be a clearance or interference depending on the desired fit. Limits and fits for inch based values are defined in ASME B4.1.

When using metric dimensions, limits of size may be calculated using the tables in ASME B4.2. Limits of size are displayed as previously explained when using limit dimensions. The largest limit is shown above the lower limit. When it is desired to indicate the limits are based on the standard tables, the limits may be specified in one of the following three ways using limits and fits codes.

When the codes are new to a company using them, metric size limits are shown and the applicable tolerance code is shown as reference.

$$\begin{array}{l} 16.043 \\ 16.000 \end{array} \left(\begin{array}{c} \\ 16H9 \end{array} \right)$$

Another allowable method is to show the limits as reference and the applicable tolerance code as the requirement.

$$16H9 \left(\begin{array}{c} 16.043 \\ 16.000 \end{array} \right)$$

It is also allowable to show only the limits and fits code. However, a drawing with tolerance codes may be confusing to users of the drawing if they do not have access to the limits and fits standard.

Size limits are calculated using either the basic hole or basic shaft system of tolerancing. The choice between the basic hole and basic shaft system is based on the design application. Either system may be utilized if all parts in the design are produced by the designing company. If one of the parts is purchased from a supplier, the choice between systems of tolerancing is dictated by the tolerances on the purchased parts.

The word *basic* when used in the context of limits of fit means something entirely different from when it is used in the context of geometric dimensioning and tolerancing as defined in Chapter 3. When used in the context of limits of fit, a basic value is one from which the limits of size are calculated. It is often incorrectly assumed that a basic size and nominal size are the same. A basic size is often equal to the nominal size, but may not be.

In regard to limits and fits, a *nominal size* is the size by which a feature is known, and that may not be an actual dimension for that feature. An example is the outside diameter for a pipe. Neither the minimum nor maximum diameter for a nominal 3/4" pipe is actually .750". Therefore, the nominal size is only a size name applied to the part. The

same is true of threads. Often, the size used to designate a thread is not one of the size limits for the thread. If it were, an internal thread and external thread would not fit together. The terms nominal and basic as defined by ASME B4.1 are only used in regard to "limits and fits" as explained in the following paragraphs.

For purposes other than limits and fits, *nominal* is generally used to mean the "as-designed" size in a CAD model or the "as-dimensioned" size to which tolerances apply.

Basic Hole System

The *basic hole system* is the preferred system for calculating limits of size. See [Figure 4-47](#). In this system, the limits of size for the hole are calculated first. Then, the limits of size for the shaft are calculated to fit the hole.

Usually, the basic size is used as the lower limit of size for the hole. The upper limit of size for the hole is determined by adding a tolerance to the lower limit. When using a standard table, the amount of tolerance is determined from the table and added to the basic size.

The limits of size for a mating shaft are calculated after the limits of size are established for the hole. The limits of size for the shaft are determined by the minimum and maximum amounts of clearance or interference that is acceptable for the given part.

The basic hole system may be used, and is preferred, when two mating parts are simultaneously designed within one company. It must be used when there is an existing hole, and a shaft is being produced to mate the predefined hole size.

It is good practice to assign standard tool dimensions as the basic size for holes. This permits the utilization of standard tools to produce the holes. If the limits of size for a hole are .125" minimum and .130" maximum, then a standard .125" diameter drill or reamer can be used to make the hole. A .125" tool is typically assumed to produce a slightly oversize hole because of imperfect tool sharpening, poor tool alignment in the machine, and other factors such as shavings scraping the sides of the hole as they travel up the flutes of the tool. However, it is also possible for a tool to produce a hole that is slightly smaller than the tool size—depending on the material and other manufacturing considerations such as tool deflections.

Basic Shaft System

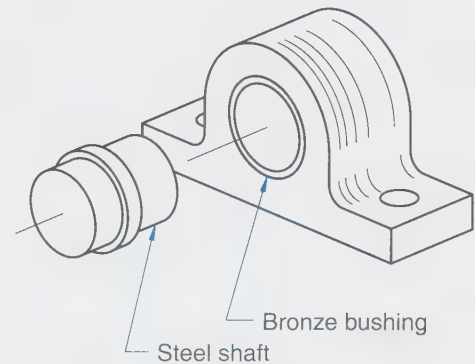
The *basic shaft system* for calculating limits of size results in assignment of the basic size as

Nominal size range, inches Over To	Class RC6		Class RC7	
	Clearance	Standard Tolerance Limits		
		Hole H8	Shaft e7	
0.40 - 0.71	1.2 3.8	+1.6 0	-1.2 -2.2	
0.71 - 1.19	1.6 4.8	+2.0 0	-1.6 -2.8	

Partial table of standard tolerance values shown in thousandths of an inch

Tolerance values are in thousandths of an inch

Ø.8750 Nominal Size
 RC6 Class fit
 .71 - 1.19 Nominal size range in tolerance table
 .0016 Minimum clearance
 .0048 Maximum clearance
 +.0020 Tolerance on hole
 -.0000 Tolerance on shaft
 -.0016 Tolerance on shaft
 -.0028 Tolerance on shaft



Basic hole calculations

Hole	Basic size	.8750	Basic size	.8750	Limits of size	Ø .8770
	Tolerance	+.0020	Tolerance	-.0000		Ø .8750
	Maximum hole	.8770	Minimum hole	.8750		
Shaft	Basic size	.8750	Basic size	.8750	Limits of size	Ø .8734
	Tolerance	-.0016	Tolerance	-.0028		Ø .8722
	Maximum shaft	.8734	Minimum shaft	.8722		

Goodheart-Willcox Publisher

Figure 4-47. Calculation of the limits of size for a specific class of fit requires that standard tolerance tables be used.

one limit dimension for the shaft. A disadvantage of this system is that the basic size of the shaft may result in calculated limits of size for the hole that do not permit utilization of standard tools. A non-standard hole size can require the use of special processes or tools to produce the hole.

The basic shaft system for calculating limits of size is typically used when the limits for the shaft must be determined first, or when the size of the shaft is predetermined. One situation requiring utilization of the basic shaft system is when the shaft is purchased, and a mating hole must be specified.

An example of when the basic shaft system must be used is when a bearing is purchased.

Limits of size for the outside diameter of the bearing are assigned by the bearing manufacturer. The bearing bore (hole) must be designed to fit the outside diameter of the bearing. The designer must determine the limits of size for the bearing bore according to the predefined bearing size.

Calculating Size Limits Using Standard Tables

Tolerances and allowances for many design applications and a wide range of nominal sizes have been standardized. These tolerances and allowances are organized into tables and categorized according to the class (type) of fit. These tables can be found in the Machinery's Handbook

and in engineering standards. For the inch system, ASME B4.1 lists the following classes of fit:

- Running and sliding clearance (RC)
- Location clearance (LC)
- Location transition (LT)
- Location interference (LN)
- Force fits (FN)

The function of the parts must be considered and a class of fit selected before the tolerance tables can be used. Each of the five general classes of fit is divided into multiple categories. Each category provides a different amount of tolerance.

Descriptions of general applications for the categories of fit are provided in Machinery's Handbook and applicable standards. These descriptions are included with the tolerance tables. A design application can be matched to one of the general descriptions to determine the class of fit to use. The appropriate table for the selected class of fit is then used to determine the tolerance values for completing the calculations.

Running and Sliding Clearance Fits

There are nine running and sliding clearance classes of fit for inch dimensions. Running or sliding clearance fits always provide a clearance between the hole and shaft. The parts move freely, but to varying degrees according to the class of running or sliding clearance (RC) fit used. Smaller RC classes of fit, such as RC1 and RC2, provide less clearance than larger ones, such as RC7, RC8, and RC9. Enough clearance is provided by the RC classes to allow lubrication.

The amount of tolerance permitted by RC classes of fit is proportional to the size of the tolerated feature. Small features are allowed smaller tolerances than large features. This is logical because a .001" clearance causes a relatively large amount of freedom on a .125" diameter shaft, but is hardly noticeable on a 2.000" diameter shaft.

Location Clearance Fits

There are eleven location clearance (LC) classes of fit. These fits are used for locating parts that must assemble without any interference between them. The LC1 class results in a very small amount of clearance, while the LC11 class results in a large amount of clearance. The smaller the clearance, the less freedom in movement there will be between the two parts. Location clearance fits provide freedom in assembly of parts, but are not intended to be used for parts that move after assembly.

Location Transition Fits

There are six location transition (LT) classes of fit. All classes can result in either a clearance or

interference. They are used for location of parts when a slight clearance or interference is acceptable. The LT1 class fit is more likely to result in a clearance than interference. The LT3 class fit has an approximately equal chance for either a clearance or interference. The LT6 class fit is more likely to result in interference than a clearance.

Location Interference Fits

There are three location interference (LN) classes of fit. These fits are used for locating parts when a clearance fit is unacceptable. A press fit ensures no freedom of movement, thus achieving an unchanging location that cannot be achieved when clearance is permitted. The LN1 class fit provides the least interference, and the LN3 class fit provides the most interference. None of the LN classes of fit are used for transmitting loads; they are only intended to be used to provide a fixed location.

Force Fits

There are five force and shrink (FN) classes of fit. All classes result in interference conditions. Each class fit provides a constant bore pressure throughout the range of nominal sizes. This means that the amount of interference between the shaft and hole varies proportionally to the basic size. The tolerances are relatively small to prevent large variations in the bore pressure.

Limit Calculations Using Tables

The class of fit to be used on a pair of parts depends on the function of the parts. When the function is known, a selection of the class of fit can be made. The selected class of fit determines which tolerance table shall be used. The nominal sizes of the parts are also affected by the function of the parts. After a tolerance table and nominal size are selected, the limits of size for the parts can be calculated.

The example in **Figure 4-47** shows how to calculate the limits of size for a shaft and hole. The nominal size for the shaft is .8750". The inside diameter of the bushing has the same nominal size as the shaft. A class RC6 fit has been selected for use. The figure shows the appropriate portion of the table for running and sliding clearance fits.

The nominal size is located in the column of the table labeled "Nominal Size Range, Inches." The nominal size of .8750 is in the 0.71–1.19 nominal size range shown by the table. Reading across the table from the 0.71–1.19 nominal size, the applicable tolerances are located under the columns headed by "Class RC6." There are three subcolumns under the Class RC6 heading.

All values shown in the tolerance table are given in thousandths of an inch. The first subcolumn under Class RC6 shows the amount of clearance that will result from an application of the given tolerances. A nominal size of .8750" requires a minimum clearance of .0016" and a maximum clearance of .0048".

The second subcolumn shows the tolerances that are to be applied to the nominal size of the hole. A .8750" diameter hole has an RC6 tolerance of +.0020" and +.0000". Applying these tolerances to the nominal size gives limits for the hole of .8770" and .8750". One size limit for the hole is equal to the basic size because the tables are based on the basic hole system.

The RC6 subcolumn for the shaft shows a -.0016" and a -.0028" tolerance that must be applied to the nominal size. Applying these tolerances to the nominal size results in limits of size for the shaft that are .8734" and .8722".

The calculations can be checked to ensure that no mistakes have been made. The difference between the minimum hole size and the maximum shaft size must equal the minimum clearance (allowance) value given in the clearance column.

Minimum hole diameter	.8750
Maximum shaft diameter	<u>-.8734</u>
Minimum clearance	.0016

The calculated minimum clearance value equals the one shown in the table, so the maximum shaft diameter and the minimum hole diameter are correct.

The difference between the values for the minimum shaft diameter and maximum hole diameter is calculated. This difference must equal the maximum clearance value shown in the table.

Maximum hole diameter	.8770
Minimum shaft diameter	<u>-.8722</u>
Maximum clearance	.0048

Both the minimum and maximum clearance values equal the values in the table. Therefore, the calculated limits of size are correct.

Limit Calculations When One Design Exists

It may be necessary to calculate the limits of size for one part when the limits of size for the mating part are already established. This situation occurs when one of the parts is purchased from a supplier.

When calculating the limits of size for features that mate with a supplier's part, the limits of size for the supplier's part must be determined.

This is done by requesting a drawing from the supplier, or by checking dimensions in the supplier's catalog.

The maximum and minimum clearance (or interference) values are obtained from the tolerance tables and applied to the limits of size for the purchased part. This determines the limits of size for the part being designed. The following example shows how such a calculation is made.

A bearing with an outside diameter of 1.2500" is to be pressed into a housing using an FN1 class fit. The limits of size for the outside diameter of the bearing are 1.2500" and 1.2495". The tolerance table shows a minimum acceptable interference of .0003" and a maximum interference of .0013". The acceptable hole sizes are calculated as follows: Minimum hole size is calculated using the maximum shaft size and the maximum interference.

Maximum shaft	1.2500
Maximum interference	<u>-.0013</u>
Minimum hole	1.2487

Maximum hole size is calculated using the minimum shaft size and the minimum interference.

Minimum shaft	1.2495
Minimum interference	<u>-.0003</u>
Maximum hole	1.2492

Limit Calculations without Tables

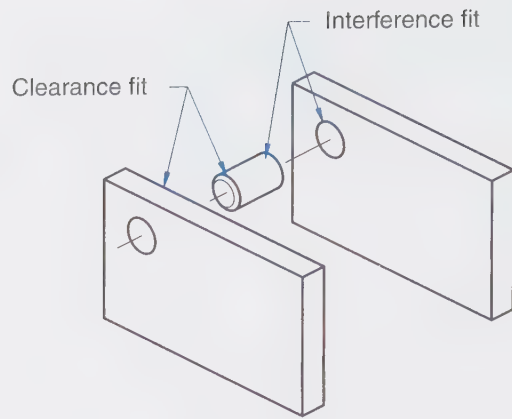
Size tolerance calculations are often made without the use of a tolerance table. See [Figure 4-48](#). In such cases, it is necessary to determine the desired allowance value and the maximum desired clearance or minimum interference. In the case of an interference fit, the acceptable bore pressure is one of the factors that should determine the maximum interference. The maximum amount of relative movement between parts is one factor used to determine the maximum clearance.

[Figure 4-48](#) shows a dowel pin that presses into one part and slips into the other. All features other than the pin holes are omitted to simplify the problem. The dowel pin limits of size are known to be .1877" and .1875". Limits of size for the two holes must be determined.

Calculation of size limits requires that the definition of allowance be understood. *Allowance* is the difference between the maximum shaft size and the minimum hole size. The allowance for a clearance fit is a positive number, and it indicates the minimum clearance between the shaft and hole. The allowance value for an interference fit is a negative number, and it indicates the maximum interference. The maximum interference may be

.1875 Nominal Size

Clearance		Interference	
Max dowel	.1877	Max dowel	.1877
Allowance	+.0010	Allowance	-.0013
Min hole		Min hole	
Min dowel	.1875	Min dowel	.1875
Max clearance	+.0030	Min interference	-.0004
Max hole		Max hole	
Clearance hole		Interference hole	
Limits of size		Limits of size	
Ø .1905		Ø .1871	
Ø .1887		Ø .1864	



Goodheart-Willcox Publisher

Figure 4-48. If the nominal size, allowance, and either a maximum clearance or minimum interference are known, the limits of size for mating parts can be calculated.

thought of as the least clearance, or the furthest condition from a clearance.

Clearance hole limits of size can only be determined after an allowance value and maximum clearance value are selected. For this example, an allowance of .0010" and a maximum clearance of .0030" are selected. The .0010" allowance is the minimum clearance between the pin and hole.

The limits of size are calculated as follows. The allowance of .0010" is added to the maximum dowel pin diameter of .1877" to determine the minimum hole diameter (.1887"). The maximum clearance value (.0030") is added to the minimum diameter of the dowel pin (.1875"). The result is a maximum hole diameter of .1905". The acceptable limits of size for the clearance hole are .1887" and .1905", which ensures a clearance of not less than .0010" and not more than .0030".

The limits of size for the interference hole in the given figure can only be determined after an allowance value (maximum interference) and a minimum interference value are selected. For this example, an allowance of .0013" and a minimum interference of .0004" are selected.

The limits are calculated as follows. The allowance (maximum interference) of .0013" is subtracted from the maximum dowel pin diameter of .1877" to determine the minimum hole diameter of .1864". The minimum interference value of .0004" is subtracted from the minimum dowel pin diameter of .1875" to determine the maximum hole diameter of .1871".

Limits of size for the interference fit hole are .1871" and .1864". These limits of size create a

minimum interference of .0004" and a maximum interference of .0013" when assembled with the dowel pin.

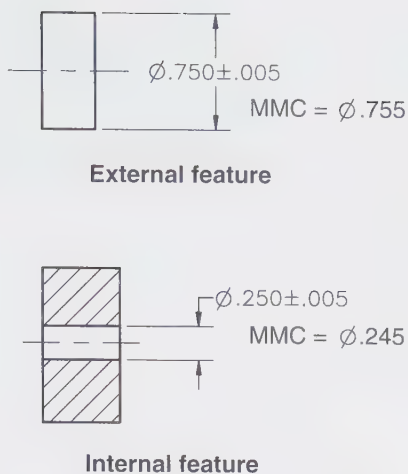
Rules in ASME Y14.5

The national standard for dimensioning and tolerancing (ASME Y14.5) contains many fundamental rules for the application of dimensions and tolerances. Only two of the rules are assigned numbers, perhaps signaling their importance. Rule #1 is essential for understanding the form requirements for individual features. Rule #2 is essential for understanding the applicable material or boundary conditions for tolerances and datum feature references.

Rule #1

The most fundamental rule that affects dimension interpretation is Rule #1. To paraphrase Rule #1, it states that any dimensioned regular feature of size must have perfect form when the feature is at its maximum material condition (MMC). The *maximum material condition* of a feature is the size at which the most material is in the part. See [Figure 4-49](#). The MMC of an external feature is equal to the maximum size limit. The MMC of an internal feature is equal to the smallest size limit.

Rule #1 requires that a regular feature of size have perfect form at MMC. This establishes what is commonly known as a *perfect form boundary* at MMC since no element of the feature may extend beyond the boundary. The standard has a different order of the words and refers to this



Goodheart-Willcox Publisher

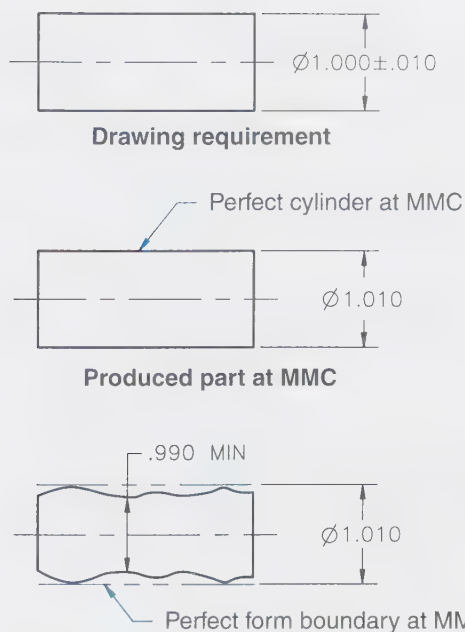
Figure 4-49. The maximum amount of material exists in the part when a dimension is at maximum material condition.

as a “boundary of perfect form.” This boundary requires that a round hole be a perfect cylinder when at its smallest permissible diameter, and that a shaft be a perfect cylinder when at its largest diameter. See [Figure 4-50](#). As a size feature departs from MMC, the feature is permitted to have form variations. The form variations are not permitted to extend beyond the perfect form boundary.

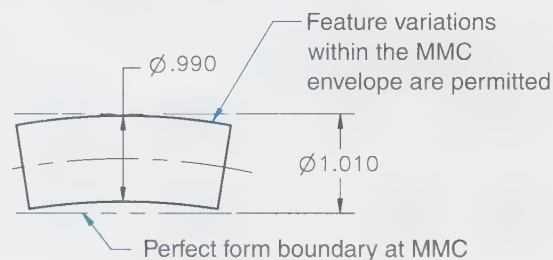
Based on the size limits alone, there is no requirement for perfect form at the least material condition (LMC). The least material condition of a feature is the size at which the least material is contained in the part. A feature produced at any acceptable size other than MMC is permitted to have form variations within the MMC boundary of perfect form. See [Figure 4-50](#). A feature produced at LMC is permitted the maximum amount of form variation.

Exceptions to Rule #1

There are some exceptions to Rule #1. The exceptions apply to parts that cannot be expected to have a perfect form. Parts that cannot be expected to have a perfect form are those subject to free state variations. This means the part is flexible to some extent. A rubber gasket may need to have a thickness dimension that is controlled by a relatively small size tolerance, but the gasket cannot be expected to stay flat when it is picked up. Another type of free state variation is the distortion of a thin-walled part that is caused by stresses built into the part. When the thin-walled part is removed from clamps that hold it during machining, the form of the part is likely to change. Rule #1 does not apply to parts that are subject to free state variation.



Produced part with size variations



Produced part at LMC

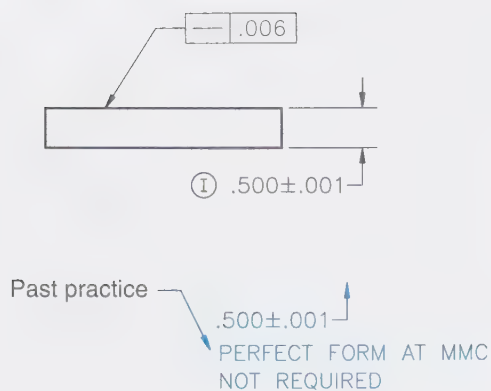
Goodheart-Willcox Publisher

Figure 4-50. A perfect form boundary exists at the maximum material condition for a regular feature of size.

The perfect form at MMC requirement does not apply to standard stock material such as bar, sheet, tube, extrusions, or other similar items. These items have standard form tolerances that permit variation of form outside the perfect form boundary at MMC.

Invoking Exceptions to Rule #1

Sometimes, the functional application for a part requires that a feature have a relatively small size tolerance, yet the functional application of that feature may permit a relatively large form tolerance. See [Figure 4-51](#). Form tolerances are covered in a later chapter, but for this explanation it is important to know that form tolerances on surfaces cannot be larger than the size tolerance that affects that surface unless an exception to Rule #1 is stated. Exception to Rule #1 is stated by showing the independency symbol adjacent to the dimension to



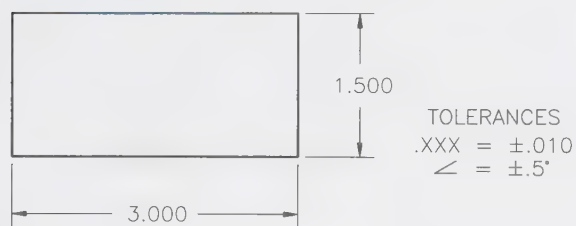
Goodheart-Willcox Publisher

Figure 4-51. Exception to Rule #1 may be invoked by placing the independency symbol adjacent to a dimension when a form tolerance exceeding the size tolerance is specified.

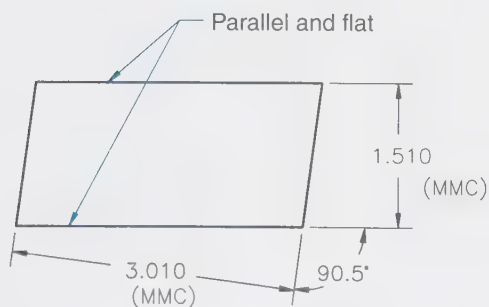
which Rule #1 is not to apply. When exception to Rule #1 is indicated with the independency symbol, a form tolerance must be shown on the feature. Past practice used a note to indicate exception to Rule #1. A note such as PERFECT FORM AT MMC NOT REQUIRED was used.

Interrelationships

Rule #1 only establishes a perfect form boundary for individual features. It does not require a perfect relationship between multiple features. See **Figure 4-52**. The given rectangle includes two



Drawing requirement



**Acceptable produced part
(exaggerated angular error)**

Goodheart-Willcox Publisher

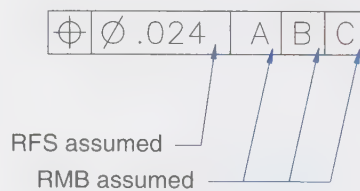
Figure 4-52. The perfect form boundary required by Rule #1 does not require perfect interrelationships between features.

features of size. They both have size tolerances of $\pm .010$ ". The 90° angle between them has a tolerance of $\pm .5^\circ$. When both features are produced at MMC, they each must have perfect form. However, there is no requirement that the 90° angles be perfect. They still have a $\pm .5^\circ$ tolerance from the noted general tolerances.

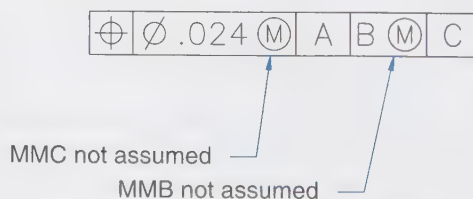
There are methods for forcing perfect interrelationships at MMC. They must not be used except when part function requires this level of control. Using unnecessary part accuracy requirements unnecessarily forces the part cost to increase. One method for forcing perfect interrelationships at MMC is to include a specific note that references dimensions to a datum reference frame. Datum reference frames are defined in the chapter on datums. Another method for forcing perfect interrelationships at MMC is to include a zero orientation tolerance at MMC. Another method would be to apply a profile tolerance all around or all over the part. Orientation and profile tolerances are explained in later chapters.

Rule #2

Rule #2 states that regardless of feature size (RFS) is assumed for all tolerances unless otherwise specified. Modifiers are applicable on the tolerance value when the associated features are features of size. If no modifier is shown following the tolerance value, RFS is assumed to apply. Material condition modifiers must be shown when a maximum material condition or least material condition is applicable to the tolerance. See **Figure 4-53**.



(M) MMC
MMB
(L) LMC
LMB



Goodheart-Willcox Publisher

Figure 4-53. RFS is assumed for all tolerances and RMB is assumed for all datum feature references.

Datum feature references are all assumed to apply regardless of material boundary (RMB) unless otherwise specified. RMB is the same as RFS except that RMB is applicable to features of size and to surfaces (non-features of size) when those surfaces have material boundaries. This is explained more completely in the chapter on datums. Datum feature references include the MMB symbol or LMB symbol when the datum feature references are to apply at the maximum material boundary (MMB) or least material boundary (LMB).

Dimensioning for Industry

Specification of size and location for each feature is the primary requirement that must be met when dimensioning, but many other factors need to be considered. An object may be dimensioned in any of several ways without violating dimensioning standards. However, in order to produce a functional part, one way of dimensioning a particular part is generally preferable. Unless the sizes and locations are specified correctly, the produced part may not assemble with its associated parts, or it may not function as intended.

Dimensions and tolerances must be applied in industry with consideration given to the part's use, associated parts, interchangeability, fabrication processes, and fabrication accuracy capabilities. These considerations affect dimension and tolerance application. Consideration and accommodation of dimension interpretation must be made for the dimension and tolerance application to be correct. The person reading the drawing or using the CAD model must interpret (read and understand) its meaning. If the interpretation is different from the true meaning, the part will not be produced as expected. Although a dimension or tolerance cannot be applied in a manner to overcome ignorance when it is interpreted, the dimension or tolerance can be applied in a manner that has one clear meaning in compliance with standard practices.

Dimensioning of a single part without knowledge of the part's function requires that size and locations be given without complete consideration of the functional relationships with associated parts. This is not an acceptable practice in industry.

After a drawing is released for production, changes to the drawing may be required for a number of reasons. Changes made after a drawing is released are known as *revisions*. Revisions to dimensions and tolerances may occur. However, keeping the number of revisions to a minimum helps to keep production costs down. The number of drawing changes can be kept to a minimum

through careful application of dimensions on the original drawing.

Interchangeability

Parts in mass-produced assemblies are normally made to be interchangeable. A design is interchangeable only if parts produced to the design documentation can be installed in place of any other part produced to the same drawing. As an example, the windshield wiper blade from a specific model car will fit any other car of the same model. Another way to look at it is that any wiper blade of the correct size is expected to fit on any car needing that size blade. Imagine if tolerances were poorly defined and every wiper blade needed custom fitting upon installation. But it is not just for high production rates that correct tolerances are important. If only one space station exists, and a part is sent into space for installation, it is good when the part can be correctly installed without rework.

The concept of interchangeable parts has existed for a long time; Eli Whitney is credited with making the first interchangeable parts in 1798. The manufacturing ability to make parts interchangeable has improved greatly over the past two centuries, and amazing accuracies are now possible.

Interchangeability can be ensured by careful analysis of the dimension tolerances to determine possible size, location, and geometry variations. One hundred percent interchangeability can be ensured by an analysis that shows the worst case conditions of all parts and their features will permit assembly of the parts. A statistical analysis may also be used to determine interchangeability or to determine risk level of failure. An explanation of tolerance calculation is given in following chapters.

Designing for interchangeability is not difficult unless the assembly is a high-precision assembly. On high-precision assemblies, tolerances may become very small, thereby causing production of parts to become more difficult or more expensive. Part of the designer's job is to find design solutions that ensure proper assembly while maximizing tolerances to keep part production as simple and inexpensive as possible, thus reducing overall costs.

It is typically preferred that parts in a design be interchangeable. Interchangeability reduces assembly time, spare parts cost, and maintenance costs.

Sometimes, the design accuracy required for interchangeable parts is not practical. In these situations, it may be best to design parts that are produced together (mated). A common type of mated

assembly is a pair of parts through which a hole is drilled. The two parts are held in the desired assembly position, and a hole is drilled through both parts. This is referred to as *mate drilling*. This process does ensure that the two holes are in line.

A problem with mate drilled holes is that a mated assembly is created. There is no assurance that parts from one assembly will fit any other assembly; therefore, interchangeability is lost. When only a few assemblies are to be made, mated parts may be preferred instead of designing interchangeable parts.

Mated assemblies reduce the number of tolerances that must be analyzed. Dimensions applied to mated features do not require a tolerance analysis because the features are machined as matched sets. However, it is not a good practice to use mated assemblies for the sole purpose of avoiding tolerance calculations.

Mated parts may initially be less expensive than interchangeable parts when only a few assemblies are produced. The initial cost is lower because design time and production tooling requirements are reduced, and machinists can work to larger tolerances. However, the assembly cost may be higher as a result of machining operations that must be performed during assembly. If the parts were interchangeable, they would simply be put together.

The life cycle cost of mated parts is not always less than interchangeable parts. Life cycle cost of mated parts must take into account the cost of maintenance. If a part breaks in a mated assembly, the entire assembly may need to be replaced, or a special replacement part must be made by transferring dimensions from the mated parts. Both alternatives for fixing the broken assembly are expensive.

Dimension Interpretations

Correct interpretation of tolerance effects is necessary to make good decisions regarding dimension application. The following paragraphs show interpretations of tolerances applied directly to angles and arcs. The accumulation of tolerances as applied directly to location dimensions is defined in the previous chapter. Explanations of how to apply geometric tolerances and also define the resulting tolerance zones are given in following chapters.

Tolerances on angled features are somewhat open to multiple interpretations unless geometric tolerances are applied. In addition to the following, it is important to also apply the practices for form and orientation tolerances that are explained in following chapters.

The tolerance zone for an angle is affected by the manner in which it is dimensioned. See [Figure 4-54](#). An angle defined with a dimension measured in degrees and having a tolerance expressed in degrees has a wedge-shaped tolerance zone. The wedge has a point at the location of the angle vertex. The vertex of the angle may be located anywhere within the zone permitted by the vertex location dimensions.

Location of the two ends of an angle create a tolerance zone for the location of each corner. Unfortunately, this method of dimensioning an angle does not provide a clear definition of the tolerance zone that exists between the two corners. The flatness of the surface is not specified. Well-defined means for specifying form and orientation tolerances are explained in the following chapters.

Radius dimensions, where not basic, have a tolerance that affects the dimensioned surface. See [Figure 4-55](#). A radius dimension of $.07'' \pm .02''$ allows any radius between $.05''$ and $.09''$. Beginning with ASME Y14.5M-1994 and continuing forward to include the current standard, the surface may lie anywhere within the boundaries. If an arc without reversals is needed, a controlled radius (CR) is specified. The given figure shows the boundaries and surface requirements for a standard radius and a controlled radius.

The interpretation of radius (R) dimension tolerances has been the subject of many standardization discussions. The radius interpretation permitting reversals of the surface became a requirement when the 1994 standard was released and continues to be applicable in current standards. The 1994 standard also introduced the controlled radius symbol. Prior to 1994, standards did not include the controlled radius symbol and the meaning of the R symbol did not permit the reversals that are now allowed.

Machine Capability

Each manufacturing machine is capable of a certain range of accuracy depending on many factors, such as the tool accuracy, machine speeds and feeds, and coolant. The tolerances applied on a drawing are going to impact the machine type and machine process selected for production of the part. See [Figure 4-56](#).

Machine processes capable of holding small tolerances are typically more expensive than similar processes that are less accurate. If tolerances of $\pm .005''$ are the smallest required in a factory, the processes are considerably less expensive than for a factory that must produce tolerances of $\pm .00005''$.

the part must be subcontracted for production by an outside company.

The surface must be at an angle between 29.5° and 30.5°

The tolerance zone widens as distance from the vertex increases

The vertex may be anywhere within the defined zone

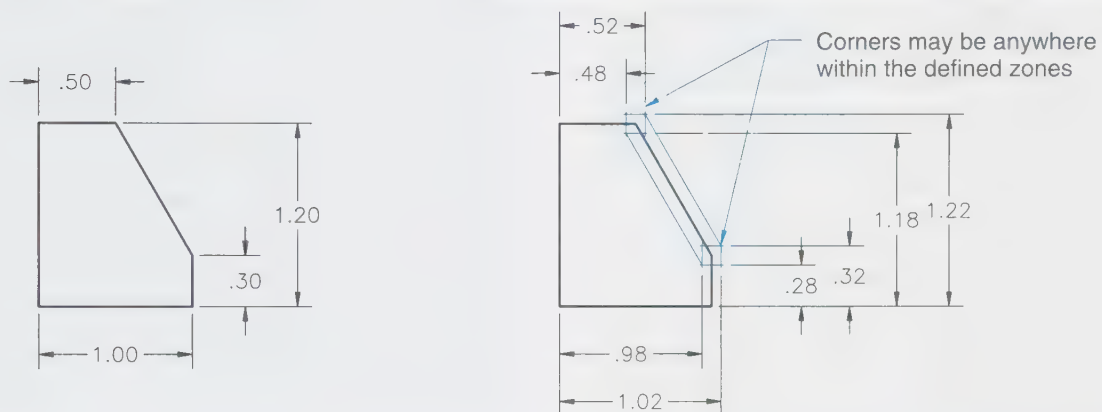


Figure 4-54. The method used to dimension an angle affects the tolerance on the angle. To more completely define the requirements, form, orientation, or profile tolerances may be used.

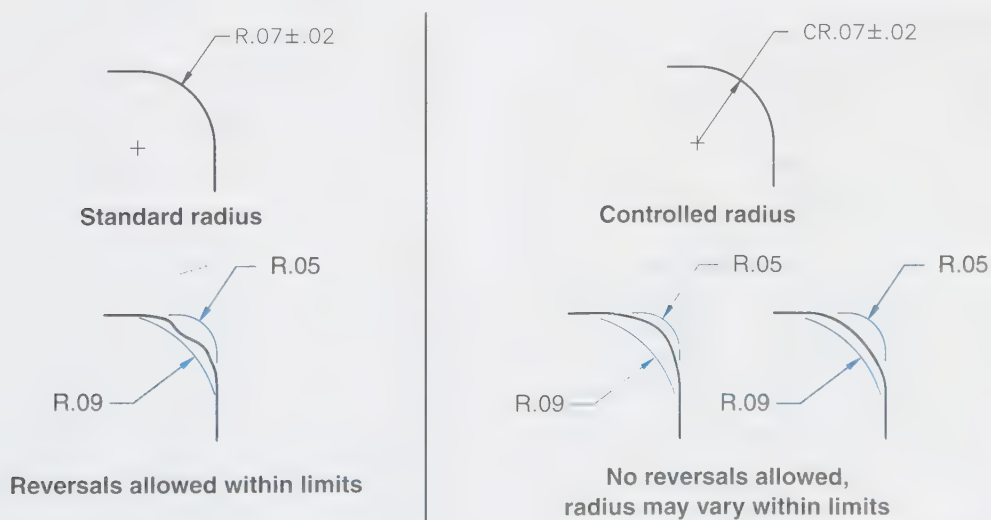


Figure 4-55. A curved surface must fall within the size limits defined by the radius dimension.

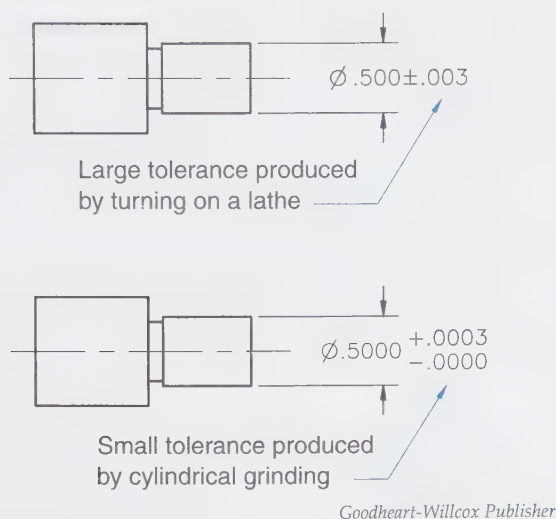


Figure 4-56. Machine capabilities must be considered when assigning tolerances.

being made. A machine that can bore a .500" diameter hole to a tolerance of .0002" in diameter may not be able to hold the same tolerance on a 6.000" diameter hole. A machine that can achieve a position tolerance of .004" diameter on a hole pattern across 12.00" may not be able to achieve the .004" diameter position tolerance across a distance of 84.00".

It should not be universally assumed that the smaller the feature, the smaller the achievable tolerance. Very small features such as holes can be difficult to produce to very small tolerances.

Machining handbooks provide information about machine process accuracies. The data in these books can provide general guidance as to the tolerance values that should be worked toward as design calculations are completed.

Dimension Revisions

Any change to a drawing original after it has been released for production must be completed through what is known as a drawing revision. Revisions may be required because of errors made on the original drawing, design changes, or changes for improving manufacturing ease.

A revision made to a dimension value on an orthographic view also requires the affected object size or location to be revised. Changing only the dimension value without changing the feature is not usually acceptable on a drawing. It is never acceptable to change a dimension in a CAD model unless the geometry is also changed to match the dimension.

In CAD models, it is extremely important to change the object geometry so that it is always to scale with the dimension. Model geometry is used

for creating computer numerical control (CNC) tool paths to run machines. If the geometry is out of scale, then parts will be made incorrectly.

Dimension revisions on a drawing graphics sheet may be made as shown in **Figure 4-57**. The revision processes vary within industry, and the figure is only meant to be an example of one way that revisions may be documented. The width of the top step on the given part is changed. The revised drawing shows the corrected front view, and the dimension is changed to show the new size. A revision letter (change identification) is placed inside a circle near the appropriate dimension to indicate that the drawing has been changed. A description of the change is entered in the revision history block on the drawing format, usually in the upper right corner of the drawing graphics sheet.

On a manually produced drawing, it is allowable for dimension changes of less than .030" to be made to the dimension value without changing the object view. Do not make this type of change to a CAD drawing. Generally, it is not a good practice to change a dimension value without also changing the object lines of the feature. If a dimension change greater than .030" is made without changing the view, the dimension is out of scale. Such a dimension must be underlined with a straight solid line to indicate that it is out of scale. See

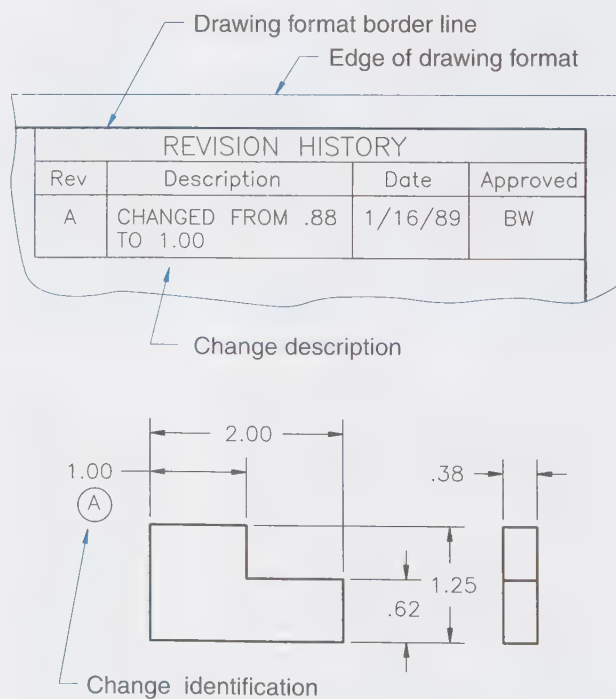


Figure 4-57. A drawing revision is normally identified where it is made and then explained in the revision history block.

Figure 4-58. This practice should only be used on manually produced drawings. It must not be used for drawings generated from solid models.

It is not a good practice to change dimensions without changing the affected features. Out-of-scale drawings become difficult to work with as additional changes are made. Out-of-scale dimensions are generally reserved for use in situations where a small change would cause a major revision to a drawing. An example is changing the location of one hole by .125" when the hole is the origin for several other features. Moving the hole would require moving all of the other features.

Pro Tip

When to Change the Geometry

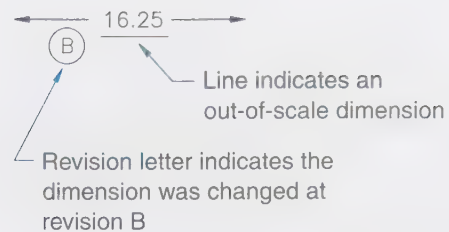
In the past, one of the decisions a drafter made was deciding whether to modify the geometry on a manual drawing or simply change the dimension. This was an understandable shortcut, considering how much time may be involved in changing a manually created drawing. With the advent of CAD, this decision is no longer made. *All geometry must match the dimension attached to it.* It is likely the model geometry will be imported into downstream applications that rely on the accuracy of the model. Examples include the machine programs that produce the parts and the inspection programs that verify dimensions are achieved.

Surface Texture Dimensions

Surface texture requirements for a part are determined according to the part's function. Examples of surface texture requirements determined by function are bearing surfaces and the outside surfaces of a gearbox. Bearings require a smooth surface finish to reduce friction. The outer surface of a gearbox housing can be relatively rough.

Surface texture is a function of the fabrication process. A sand cast part has a surface texture that is affected by the size of sand used in the mold. The texture on a lathe-turned part is affected by the cutting tool, machine speed, and feed. The texture on a ground part is smooth because of the small grain size in the grinding wheel, the high speed of the wheel, and a relatively slow feed.

Specification of tolerances on dimensions has an indirect effect on surface texture. As tolerances become small, the fabrication processes become more exacting. More exacting fabrication processes result in better quality surface textures. A



Goodheart-Willcox Publisher

Figure 4-58. Out-of-scale dimensions should always be avoided and generally are used only when making a drawing revision on manually produced drawings. It should never be done in CAD-based models.

tolerance of ± 0.0002 " on a shaft diameter forces the part to be ground or lapped. Grinding to achieve a tolerance this small will probably result in a surface finish of 32 microinches or better. (A micro-inch is .000001".) In effect, the ± 0.0002 " size tolerance forced a good-quality surface texture.

Surface texture may be specified by showing surface control symbols applied to the profile in an orthographic view of the controlled surface. This is done if the size tolerance is not certain to result in the desired texture. The surface texture symbol is applied where the surface appears as a line. Generally, the minimum information specified will include a surface roughness value. Surface roughness is interpreted in accordance with ASME B46.1, and applied per ASME Y14.36.

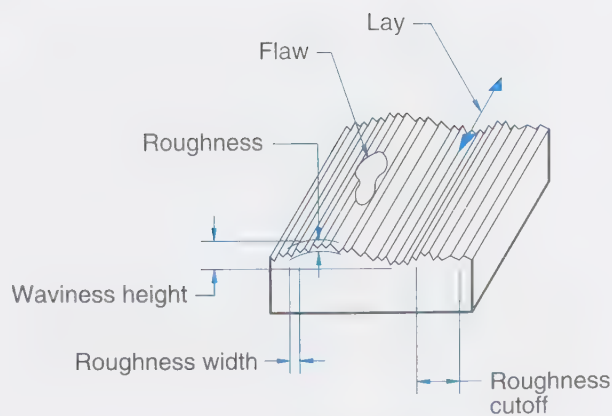
Manufacturing costs increase as surface control increases. Manufacturing methods can be dictated by the degree of surface control specified. In order to keep costs down, surface texture must not be overspecified. Care in applying surface texture must be exercised to prevent specifying a machine process that is more restricting than is required by the design function.

Surface conditions include variations known as roughness, waviness, and lay. Surface **roughness** is the height of small peaks and valleys on a surface. See **Figure 4-59**. **Waviness** is a larger variation in surface texture than roughness. Roughness variations are superimposed on the waviness variations. **Lay** is the direction of tool marks, scratches, or the grain that affects surface texture.

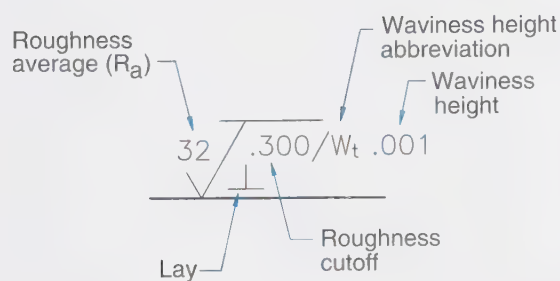
Surface texture symbols are drawn as shown in the given figure. The horizontal bar can be omitted from the symbol when surface texture values are not specified. The symbol is not to be placed so that it is inclined or upside down.

Surface Roughness

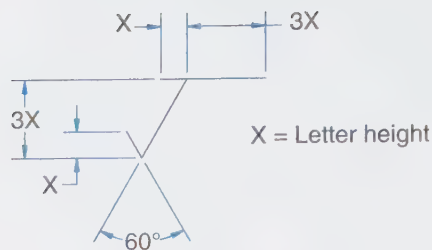
Surface roughness and roughness cutoff are often the only numerical values specified. The surface texture symbol, for roughness control, will



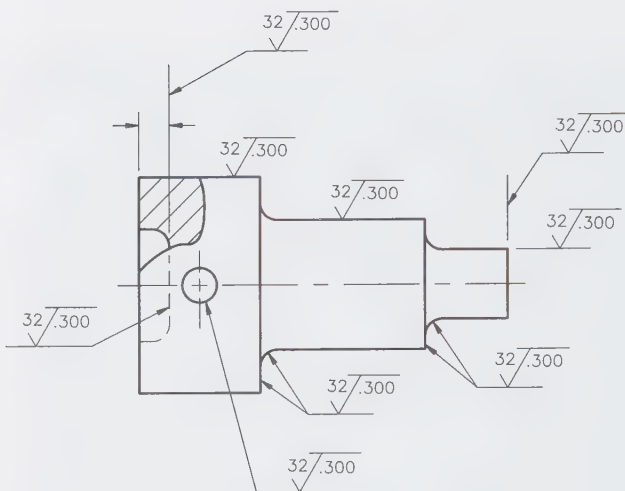
Terminology



Surface texture specification



Surface texture symbol



Applied symbols

Goodheart-Willcox Publisher

Figure 4-59. Surface texture symbols specify texture requirements.

also include a roughness width cutoff value if compliant with ASME Y14.36M-1996. See [Figure 4-60](#). The average roughness value is shown in the "V." Microinches are specified if the inch system is being used. Micrometers are specified if the metric system is in use. Standard values for surface roughness are listed in the given figure. If only one roughness value is shown in the symbol, it is the maximum acceptable roughness. Two roughness values indicate a maximum and minimum acceptable roughness. The roughness value represents the arithmetic average deviation from a mean line on a surface profile.

A general note may be used to indicate the surface roughness requirement for all surfaces except those with a symbol applied to them.

Surface roughness can be measured by electronic inspection equipment or by comparison to sample surfaces of known roughness. The comparison method is less accurate, but it is also less expensive. If roughness requirements are not very demanding, the comparison method may be preferred.

The distance across peaks and valleys that cause surface roughness is known as the *roughness width*. If required, a maximum width may be noted.

The distance across which roughness values are measured is commonly called the *roughness width cutoff*, *roughness cutoff*, or *sampling length*. The roughness width cutoff distance must be specified in documents created since 1995. Prior to 1995, a roughness width cutoff of 0.8 mm was the default. The value is placed directly under the horizontal bar of the surface texture symbol. Commonly used roughness cutoff values are shown in the given figure.

Waviness

Waviness is the large surface variations on which surface roughness is superimposed. See [Figure 4-61](#). These variations are expressed in decimal parts of an inch or millimeters. Common values used for waviness are shown in the given figure. Waviness values are placed just right of the cutoff rating.

Microinches

$$32 \sqrt{\quad} .100$$

Maximum roughness and
roughness cutoff

Micrometers

$$0.8 \sqrt{\quad} 2.5$$

Roughness width

Roughness

Roughness cutoff

$$63 \sqrt{\quad} .100$$

Minimum and maximum roughness
and roughness cutoff

$$1.6 \sqrt{\quad} 2.5$$

Roughness Cutoff Values

Inch	Millimeter
.003	0.08
.013	0.25
.030	0.80
.100	2.50
.300	8.00
1.000	25.00

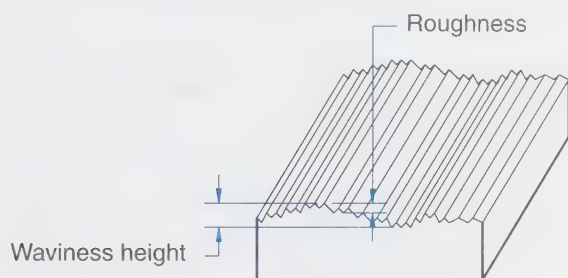
Values shown are not
conversions

Roughness Average Values

Microinch	Micrometer
2	0.05
4	0.10
8	0.20
16	0.40
32	0.80
63	1.60
125	3.20
250	6.30
500	12.50

Goodheart-Willcox Publisher

Figure 4-60. Only the required level of control for surface finish is specified.



Microinches

$$32 \sqrt{\quad} .300/W_t .001$$

Micrometers

$$0.8 \sqrt{\quad} 2.5/W_t .025$$

Roughness, roughness cutoff, and waviness height

Common Waviness
Height Values

Inch	Millimeter
.00002	0.0005
.00005	0.0012
.0001	0.0025
.0002	0.005
.0005	0.012
.001	0.025
.002	0.050
.005	0.12
.01	0.25
.02	0.50

Goodheart-Willcox Publisher

Figure 4-61. Roughness variations may fall within larger surface variations known as waviness.

Waviness is not meant to be a replacement for the flatness tolerance explained in the chapter on form tolerances. If a waviness tolerance value is shown, it should always be less than any flatness tolerance that is on the same surface.

Process

When a specific process is required, it may be noted above the horizontal line of the surface texture symbol.

Lay

Lay is the direction of surface lines caused by cutting tools. The lay direction can have an effect on the function of a part. A bearing is used to reduce friction in a machine; the lay on a bearing surface has a definite effect on friction. Specification of lay on a bearing surface ensures control of the amount of friction that will be present.

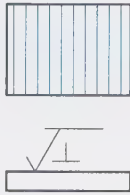
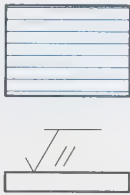
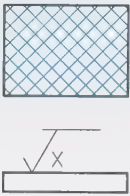
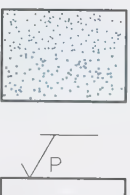
Lay is specified on a drawing by showing a lay direction symbol in the surface texture symbol. See

Figure 4-62 for an example. The lay symbol indicates the direction of lay relative to the profile view on which the symbol is shown. Lay must always be indicated on a profile view of the affected surface.

Parallel and perpendicular lay paths are caused by straight cutter motions. Angular paths are generated by straight and large radius cutter motion. Circular lay is caused by operations such as facing parts on a lathe. Multidirectional lay can be caused by an end mill. Radial lay is caused by some grinding operations. Particulate lay is caused by powder metallurgy, casting, and similar processes.

Past Practices

The material in this chapter reflects practices that are in the current standard except where

Symbol	Application	Description
⊥		Perpendicular
//		Parallel
X		Angular in two directions
P		Particulate, nondirectional, or protuberant



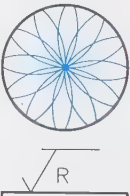
noted otherwise. Because these practices developed over time, many drawings exist that were created in compliance with previous editions of the national standards. Some of the commonly seen past practices related to the content of this chapter are explained below.

Diameter Symbol

Prior to the 1982 standard, there was no diameter symbol defined and the abbreviation DIA was applied to diameter dimensions. The abbreviation followed the dimension value.

Finish Symbols

It was common practice to indicate machined (or finished) surfaces on castings, forgings, and

Symbol	Application	Description
M		Multidirectional
C		Circular
R		Radial



Goodheart-Willcox Publisher

Figure 4-62. Lay is the direction of surface lines caused by the path of the cutting tools or fabrication process.

other part types that included secondary machining operations. Two symbols have been used to indicate finish surfaces. The most recent symbol was shaped like a letter V. The sides of the symbol formed a 60° angle. The vertex was placed on the finished surface where the surface appeared as an object line. When the finish symbol could not be placed on the surface, it was placed on an extension line from the surface. The symbol size was .12". The symbol was placed on an object line or extension line so that it pointed toward the machined surface.

Another symbol looks somewhat like a lower-case letter F with the symbol applied crossing the object line or across an extension line from the finished surface. The *f* symbol was meant to indicate the surface was "finished."

Perfect Form at MMC Not Required

Prior to the independency symbol introduction, an exception to Rule #1 was indicated by a notation placed adjacent to the applicable dimension value. The note was typically something that indicated the PERFECT FORM AT MMC IS NOT REQUIRED.

Alternate to Rule #2

In past practice, a letter S inside a circle was the regardless of feature size symbol and was used in positional tolerances when applicable. The ASME Y14.5-1982 standard required the RFS symbol in position tolerances when the tolerance or datum feature references were applicable on an RFS basis. The ASME Y14.5M-1994 standard permitted its use in position tolerances as alternate Rule #2B, but its use was not required.

Chapter Summary

- ✓ Generally, a regular feature of size must have perfect form when at the maximum material condition.
- ✓ Every feature of size must have some amount of permissible variation in its size.
- ✓ Size tolerances are calculated using either the basic hole or basic shaft system.
- ✓ Standard tables provide the allowances and tolerances required for classes of fit that meet many common design applications.
- ✓ There are two essential rules in the current dimensioning and tolerancing standard. These rules define the requirement for perfect form at MMC and the requirements for applicable modifiers in feature control frames.

- ✓ Interchangeability is an important reason for properly applying dimensions and tolerances.
- ✓ Surface control specifications are used only to the extent necessary for the part function.
- ✓ Surface roughness on a drawing is specified in microinches or micrometers.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. The current dimensioning standard requires that diameter dimensions be ____.
 - A. preceded by a diameter symbol
 - B. followed by a diameter symbol
 - C. preceded by the diameter abbreviation
 - D. followed by the diameter abbreviation
2. A flat on a shaft must be dimensioned by specifying the distance from the ____.
 - A. centerline to the flat
 - B. far side of the shaft to the flat
 - C. top of the removed portion of the shaft to the flat
 - D. Either A or B.
3. A dimension showing the location of a round hole must be applied to ____.
 - A. the center of the hole
 - B. the edge of the hole
 - C. provide two methods for determining location
 - D. Either A or B.
4. The depth of a hole is the distance into the part that the ____ extends.
 - A. drill point
 - B. full diameter
 - C. centerline
 - D. None of the above.
5. A counterbore depth is controlled by specifying ____.
 - A. remaining material
 - B. depth from the surface
 - C. noted depth from the surface
 - D. Any of the above.

6. The specified angle of a countersink is the _____.
 - A. included angle
 - B. angle between the centerline and countersink surface
 - C. angle between the part surface and the countersink surface
 - D. None of the above.
7. An angle dimension includes a dimension line that is drawn with an arc. The center for the arc is located at the _____.
 - A. edge on the part
 - B. vertex of the angle
 - C. location that makes the arc appear correct
 - D. midpoint between the part and the dimension line location
8. The leader used to show a radius dimension must point at or pass through _____.
 - A. one tangent point
 - B. a location dimension
 - C. the arc center point
 - D. Either A or C.
9. The arc length may be dimensioned by using an arc to draw the dimension line and placing a(n) _____.
 - A. straight line above the dimension
 - B. arc above the dimension
 - C. straight line below the dimension
 - D. arc below the dimension
10. A dimension showing the location of a sheet metal bend applies to the ____ location.
 - A. mold line
 - B. neutral line
 - C. bend centerline
 - D. bend tangent point
11. Based on past practices, an existing drawing may include a symbol that looks like the letter V to indicate _____.
 - A. critical surfaces
 - B. cast surfaces
 - C. critical dimensions
 - D. finished surfaces
12. Rule #1 requires that a part have perfect form when at _____.
 - A. MMC
 - B. LMC
 - C. RFS
 - D. Both A and B.
13. Two features shown at a 90° angle on a drawing are both produced at the size limit at which perfect form is required. The 90° angle _____.
 - A. must be a perfect 90°
 - B. can vary by the angular tolerance for the 90° dimension
 - C. can vary only within the perfect form boundary
 - D. Inadequate information given.
14. On all geometric tolerances, _____.
 - A. LMC is assumed to apply unless otherwise specified
 - B. RFS is assumed to apply unless otherwise specified
 - C. MMC is assumed to apply unless otherwise specified
 - D. material condition modifiers must be shown
15. The limits for a radius dimension permit the arc to vary in radius, _____.
 - A. but no flats are permitted on the surface
 - B. but no reversals are permitted on the surface
 - C. and the form of the surface may include reversals
 - D. Both A and B.
16. The tolerances applied to a feature affect _____.
 - A. which machine processes can be used to produce the part
 - B. the cost of the part
 - C. how well the part meets its functional requirements
 - D. All of the above.
17. The ____ indicates the required pattern for the surface roughness.
 - A. roughness value
 - B. waviness value
 - C. roughness width
 - D. lay direction

True/False

18. *True or False?* The height, width, and depth must be given when dimensioning a prism.
19. *True or False?* A diameter may only be specified when the feature is a complete circle.
20. *True or False?* A counterdrill specification may include the angle at the bottom of the counterdrill.

21. *True or False?* A 30° chamfer may be dimensioned with a note.
22. *True or False?* The taper on a shaft or hole may be dimensioned by specifying the amount of change in the diameter over a given length of the part.
23. *True or False?* In an orthographic view, the center point for an arc must never be moved from its true location for dimensioning purposes.
24. *True or False?* A spherical radius has a different prefix (symbol) than a nonspherical radius.
25. *True or False?* The true size of a feature must be dimensioned even if a segment of the feature is removed to reduce the size of an orthographic view.
26. *True or False?* An angle tolerance specified in degrees, minutes, and seconds is wedge shaped.
27. *True or False?* Dimensions define the part requirements such as hole diameter, and typically include instructions as to which machine processes are required.
28. *True or False?* Dimension text in a CAD model should never be edited without changing the model geometry to accurately reflect the dimension numbers.
36. Under what conditions might it be necessary to draw offsets in extension lines?
37. List three methods of giving the acceptable size limits for a dimension.
38. How is exception to Rule #1 taken when a form tolerance greater than the size tolerance is to be specified?
39. List the two measurement units that are used to measure surface roughness.
40. Complete a specification for a counterbored hole that meets the following requirements. Use symbology in the specification.
 - Hole diameter = .250"
 - Counterbore diameter = .375"
 - Counterbore depth = .250"
 - Fillet radius = .030"
41. Complete a specification for a countersunk hole that meets the following requirements. Use symbology in the specification.
 - Hole diameter = .375"
 - Countersink diameter = .625"
 - Countersink angle = 100°

Fill in the Blank

29. When dimensioning a cylinder, the _____ and length must be specified.
30. An arc is dimensioned by specifying its _____ and location.
31. The prefix used for a spherical radius is _____.
32. A _____ symbol may be used to specify surface roughness, waviness, and other surface irregularities.

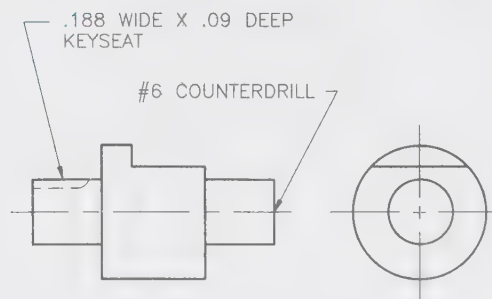
Short Answer

33. List the information that must be provided in a counterbore specification.
34. List two methods for providing the necessary definition for the location of an arc.
35. Explain how the line of symmetry is shown on a drawing.

Application Problems

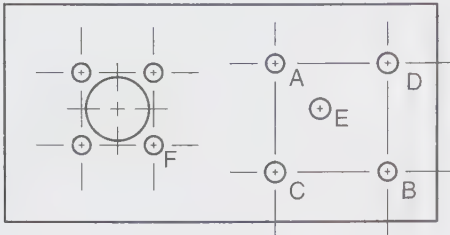
Each of the following problems requires that a sketch be completed. All sketches should be neat and accurate. Dimensions must be applied to the sketch. Unspecified dimensions may be approximated using the shown grid. Dimensions for approximated sizes and locations will be evaluated on the basis of how the dimension is applied, not on the accuracy of the approximation. Some of the following problems require calculations. Dimensions applied on the basis of calculations must be the correct number values.

42. The views in the following problem are incomplete. Make a sketch of the required views and apply all dimensions.



Grid spacing = .250

43. Sketch the shown object and apply all dimensions. Hole sizes provided in the table are to be applied using leader lines and notations. Either the inch or metric part may be completed.

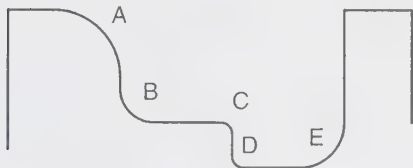


Grid spacing = .250
or
Grid spacing = 10.0 mm

Inch and metric parts are different sizes. Values given are not conversions.

Hole Table		
Hole	Inch	Millimeter
A	.250	9.0
B	.250	9.0
C	.280	10.0
D	.280	10.0
E	.280	10.0
F	.188	8.0

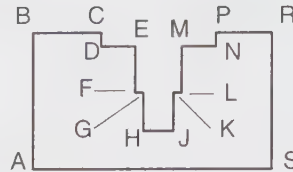
44. Apply the radius dimensions to the given figure. Either the inch or metric part may be completed.



Radius	A	B	C	D	E
Inch	.750	.375	.125	.125	.500
Millimeter	30.0	20.0	5.0	5.0	20.0

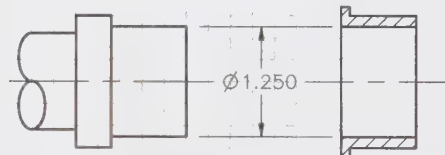
Grid spacing = .250
or
Grid spacing = 10 mm

45. Complete a drawing of the following part at a scale of 1/1. Do not increase the scale. Apply all dimensions. Joggles may be placed in the extension lines, if needed. Do not use rectangular coordinate dimensioning without dimension lines.



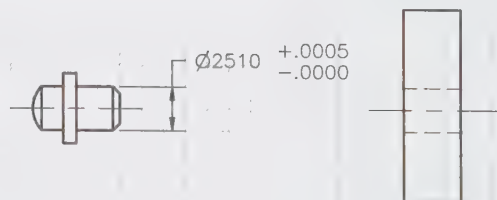
Hole Table		
Point	X	Y
A	0	0
B	0	1.00
C	.50	1.00
D	.50	.90
E	.75	.90
F	.75	.56
G	.81	.56
H	.81	.28
J	1.03	.28
K	1.03	.56
L	1.09	.56
M	1.09	.90
N	1.34	.90
P	1.34	1.00
R	1.75	1.00
S	1.75	0

46. Completely dimension the two parts. The 1.250" nominal diameter features are to be dimensioned for an RC7 class fit. Tolerance values for an RC7 class fit can be obtained from engineering standards or a Machinery's Handbook.



Grid spacing = .250

47. Dimension the hole to result in a .0025" maximum and .0010" minimum interference with the shown rest button.



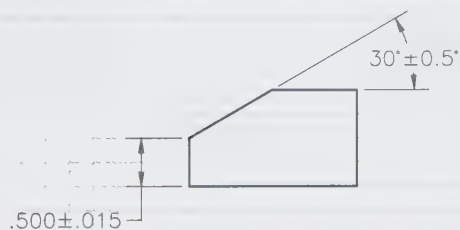
Grid spacing = .125

48. Show one acceptable distorted shape for the given part when all diameter measurements on a produced part are at 2.48" (LMC). Show whether the part is within the MMC boundary.



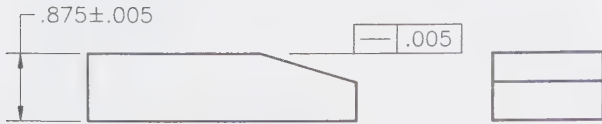
Grid spacing = 1.00

49. Show two acceptable extremes for the angled surface with the vertex produced at a location of .485" from the bottom surface.



Grid spacing = .250

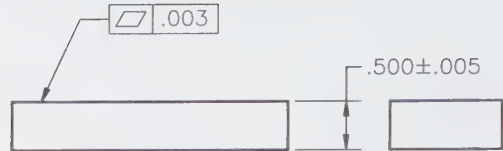
— .005



Straightness

Goodheart-Willcox Publisher

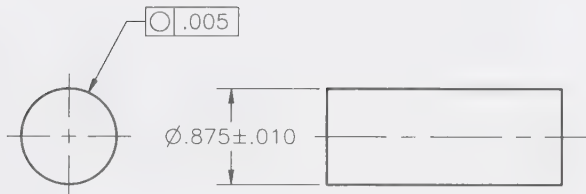
▱ .003



Flatness

Goodheart-Willcox Publisher

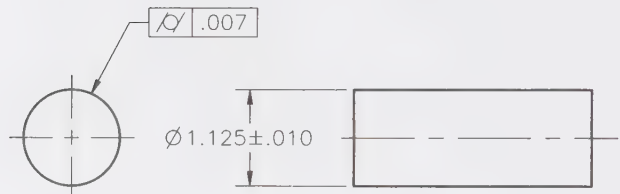
○ .005



Circularity

Goodheart-Willcox Publisher

⌀ .007



Cylindricity

Goodheart-Willcox Publisher

Chapter 5

Form Tolerances

Objectives

Information in this chapter will enable you to:

- ▼ Draw the symbols for form tolerances.
- ▼ Complete a feature control frame to specify a form tolerance and properly apply material condition modifiers on the tolerances.
- ▼ Explain the extent of form control established by limits of size.
- ▼ Apply straightness tolerances to control surface elements or a derived median line and show the interpretation of those tolerances.
- ▼ Explain and calculate virtual condition for a regular feature of size that has a form tolerance applied to it.
- ▼ Apply flatness to control a surface and show an interpretation of the flatness tolerance zone.
- ▼ Apply a flatness tolerance for a median plane and show an interpretation of the flatness tolerance zone.
- ▼ Apply circularity tolerances and show an interpretation of a circularity tolerance zone.
- ▼ Apply a cylindricity tolerance and show an interpretation of the cylindricity tolerance zone.

Technical Terms

actual mating envelope
bonus tolerance
center line
circularity
coordinate measuring machines (CMM)
cylindricity
derived median line
envelope principle
feature control frame

flatness
free state variation
full indicator movement (FIM)
functional gages
least material condition (LMC)
maximum material condition (MMC)
Rule #1
sine bar
straightness
surface line element
tolerance symbol
tolerance value
total indicator reading (TIR)
virtual condition

Introduction

All parts include shape and surface variations. The allowable amount of form variation on individual features is often limited by size tolerances. Further definition of allowable form variation on individual features may be specified through the application of form tolerances as explained in this chapter. Allowable form variation may also be specified using profile, orientation, and runout tolerances as explained in other chapters.

A form tolerance is needed when other tolerances such as size, profile, orientation, or runout do not provide adequate control of a feature. Tolerances associated with size dimensions permit form variations equal to the size tolerance. If the form variations permitted by the size tolerances are acceptable, then no additional form tolerance specification is needed. Similarly, if a profile, orientation, or runout tolerance adequately controls the form, then no additional form tolerance is required. Only when the form variations must be less than is

permitted by the size, orientation, profile, or runout tolerances is it necessary to specify a form tolerance.

If size, profile, orientation, and runout tolerances establish a form requirement, why is there a special category of tolerances called form tolerances? Form tolerances provide the ability to specify the needed form precision without requiring overly restrictive size, orientation, profile, or runout tolerances. As an example, a relatively large size tolerance may be allowable, but the shape (form) of the feature must be controlled to a very small value. A specific application of this type is a surface plate used for inspection. It may have a relatively large thickness tolerance, but the flatness tolerance on the surface must be very small.

Form tolerances control variations on individual features. Applying a flatness tolerance on a single surface only requires control of that particular surface. Form tolerances do not control the relationship (angle or location) between features. A flatness form tolerance, for example, does not imply a parallelism or perpendicularity requirement relative to other features. Furthermore, a form tolerance applied to two or more adjacent surfaces only controls flatness of each surface independent of the others. It does not control coplanarity of the adjacent surfaces.

Because a size tolerance limits the allowable form variation, it is true that a smaller size tolerance reduces the permitted form variations. From this it may seem that form tolerances could be eliminated by using smaller size tolerances to achieve the desired form control. However, controlling form variations through excessively small size tolerances has the potential to unnecessarily increase part cost. Also, many surfaces are not defined by size dimensions, so the form of those surfaces is not affected by a size dimension.

The use of a form tolerance is desirable when the needed form quality is smaller than the other allowable geometric variations, because it is usually easier to control the form of a surface, such as flatness, than it is to control size, orientation, or location. As an example, part function may require a surface that is flat within .004" and also allow a relatively large $\pm .030$ " size (thickness) tolerance. The following paragraph explains how these two requirements may be specified.

One option for meeting the functional requirement for flatness of .004" is to apply a small size tolerance of $\pm .002$ " to the thickness. The size dimension establishes both the thickness and flatness requirements using one size tolerance value. Unfortunately, this approach requires the thickness

to be controlled to a value only 7% of the .060" size range that could be permitted. A second and better option is to specify the needed flatness tolerance and maximize the allowable size tolerance. A flatness tolerance of .004" and a separate $\pm .030$ " size (thickness) tolerance may be specified. This approach meets functional requirements, ensures the greatest manufacturing ease, and provides the lowest cost part. Correct application of form tolerances can, on some parts, permit other tolerances to be increased and thereby reduce fabrication cost while meeting functional requirements. A part that has a flatness tolerance specification of .004" and a size (thickness) tolerance of $\pm .030$ " can be produced at a relatively low cost when compared to one that has a size tolerance of $\pm .002$ ".

Size tolerances should be used to define the allowable size variations of a part. If the needed size tolerance adequately controls the form, there is no need to add a form tolerance. However, if form variations need to be controlled to a value less than the size tolerance, form tolerances need to be specified so that size tolerances are not made overly restrictive. Part cost can be minimized by properly utilizing size and form tolerances to define the needed product quality without making any of the tolerances smaller than needed.

Form Tolerance Categories

The form tolerances are straightness, flatness, circularity, and cylindricity. Each form tolerance has a specific definition. Application of the proper tolerance specification requires understanding the possible uses for the form tolerances.

Straightness may include a material condition modifier when it is applied to control the derived median line of a cylindrical feature, and flatness may include a material condition modifier when it is applied to control a derived median plane. Material condition modifiers were introduced in Chapter 2 and defined to a greater extent in Chapters 3 and 4. Appropriate applications of the material condition modifiers and how they impact straightness and flatness tolerances are explained in this chapter.

Form Tolerance Symbols

Symbols used for form tolerances are easy to remember because their shapes are symbolic of the indicated control. See [Figure 5-1](#). A straight line indicates straightness. A parallelogram represents flatness. A circle stands for circularity, and a circle with tangent lines means cylindricity.

—	Straightness
▭	Flatness
○	Circularity
⊘	Cylindricity

Goodheart-Willcox Publisher

Figure 5-1. There are four form tolerance symbols.

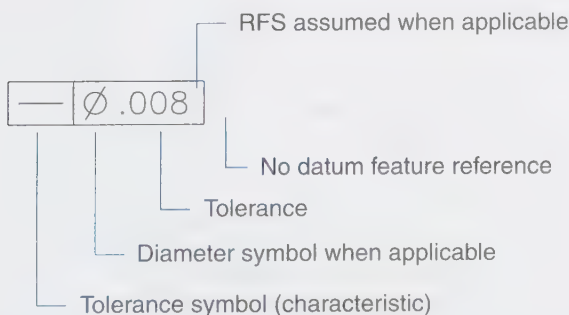
Feature Control Frames

Proper format and application of the tolerance specification is necessary to convey the correct information. See **Figure 5-2**. The proper specification format, when using symbology, requires the tolerance be shown in a *feature control frame*. Feature control frames must be organized as shown in the given figure. The manner in which the feature control frame is shown on the drawing or in a model affects the meaning of the tolerance.

The feature control frame for a form tolerance only has two compartments. The first compartment always contains the *tolerance symbol*. The second compartment contains the *tolerance value* that indicates the amount of acceptable tolerance. Depending on the tolerance being specified, there may be a diameter symbol in front of the tolerance value. A material condition modifier is applicable when straightness or flatness is applied to a feature of size, and the applicable material condition may be specified as RFS, MMC, or LMC. RFS applies when no symbol is shown, because RFS is assumed. To specify that MMC or LMC is applicable, the MMC or LMC symbol is placed following the tolerance value.

The statistical symbol and the free state symbol are shown when applicable. These symbols are placed after any applicable material condition modifier.

A datum feature reference is never included in a form tolerance specification for the purpose



Goodheart-Willcox Publisher

Figure 5-2. A feature control frame for a form tolerance always contains a tolerance symbol and a tolerance value.

of constraining translation or rotation of the feature relative to the datum. It would be incorrect to use a datum feature reference in form tolerances because these tolerances are used for the control of individual features and do not include any requirement for orientation or location relationships between features.

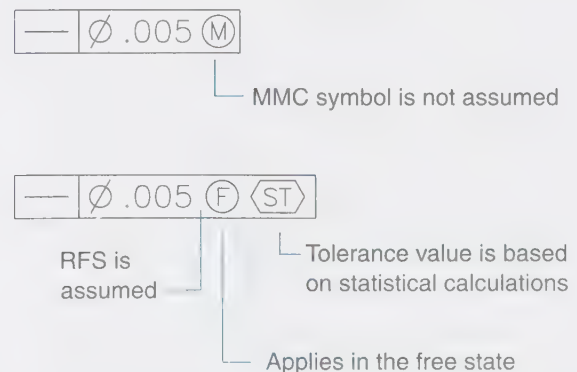
Material Condition Modifiers

Straightness and flatness tolerances applied to a regular feature of size require that a material condition modifier be associated with the specified tolerance value. The maximum material condition (MMC) modifier must be shown in the feature control frame if it is to apply. See **Figure 5-3**. If no material condition modifier is shown in the feature control frame, then the tolerance is understood (assumed) to apply regardless of feature size (RFS). This is in compliance with Rule #2 of the current national standard.

Care must be taken to show the MMC modifier in all situations where the tolerance should apply at MMC. Omission of the MMC modifier will result in less available tolerance and could potentially increase part cost. Just as importantly, the MMC modifier must not be shown if the tolerance should apply regardless of feature size (RFS). Application of MMC or RFS in a form tolerance specification causes significant differences in the resulting part requirements.

Application of the LMC modifier on a straightness or flatness tolerance is seldom necessary. If needed, the LMC modifier is shown in the feature control frame following the tolerance value.

Material condition modifiers are only used when the form tolerance is applied to a regular feature of size. Application of a form tolerance to surface elements does not require the application of a



Goodheart-Willcox Publisher

Figure 5-3. The material condition modifier on a form tolerance is assumed to be RFS unless otherwise shown.

material condition modifier. In fact, it is incorrect to show a material condition modifier on a form tolerance applied to surface elements.

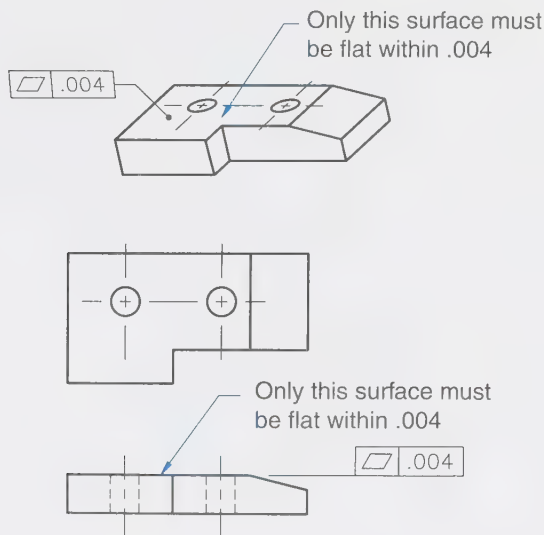
Form Tolerance Applicable to Single Features

Each form tolerance only affects the single feature to which it is applied. See [Figure 5-4](#). A flatness tolerance applied to the top surface of a part only expresses a requirement for that surface. If control of another surface is desired, another feature control frame may be drawn. It is also possible that two leaders may extend from one feature control frame to two surfaces. If this is done, it is the same as two separate feature control frames on the drawing and does not imply that the form tolerance establishes a relationship between the two features.

When a form tolerance is applied to an interrupted feature of size, the form tolerance is understood to apply to all the surface segments as if it was one feature when the continuous feature symbol is applied. Use of the continuous feature symbol is only recommended for features of size and is not recommended for surfaces.

Surfaces and Features of Size

It is extremely important that the difference between surfaces and features of size be understood. The method of application of a feature control frame to a part determines whether the tolerance is applicable to a surface or a regular feature of size. A feature control frame has a significantly different meaning depending on whether it is attached to a surface or a feature of size.

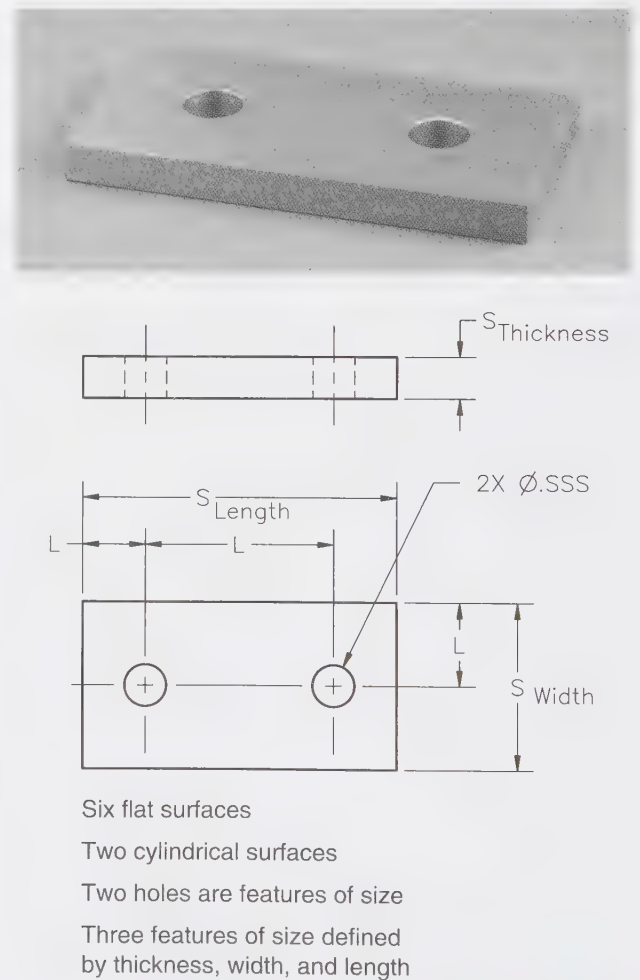


Goodheart-Willcox Publisher

The difference between surfaces and regular features of size is shown in [Figure 5-5](#). In the given figure, regular features of size are those defined by size dimensions indicated by the letter S. The given rectangular part includes two holes. The hole diameters are defined by size dimensions. The holes are therefore features of size. Each hole also has a cylindrical surface.

Three regular features of size are defined by the thickness, width, and length of the part. Each feature of size is made of two surfaces separated by a size dimension. There are six flat surfaces on the part. Each of these, when considered independently of all other features, is a surface.

There are two classes of features of size. Regular features of size are typically associated with a single size dimension such as thickness, width, length, or diameter as shown in [Figure 5-5](#). Irregular features of size may be one enclosed shape or a collection of features that in combination establishes a size boundary.



Goodheart-Willcox Publisher

Figure 5-5. The difference between surfaces and features of size must be understood to properly apply form tolerances.

Form Requirements from Size Limits

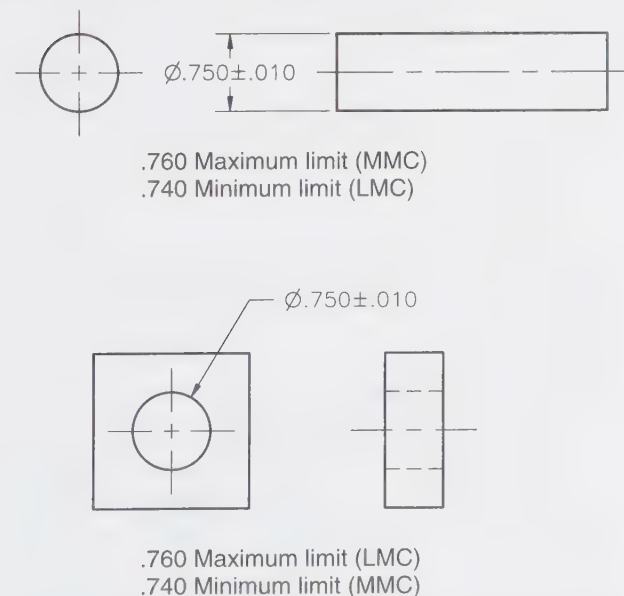
Every regular feature of size must have a size dimension, and every size dimension has a size tolerance. The size tolerance is directly applied to the size dimension or applied as a profile tolerance. The size tolerance applied to a regular feature of size creates a maximum and minimum size limit. The maximum size, the minimum size, and form variation that may exist on a feature are controlled by the limits of size.

Variations in size have an effect on the amount of material in a part. See **Figure 5-6**. The **maximum material condition (MMC)** is when the maximum permissible amount of material exists. The **least material condition (LMC)** is when the least amount of material exists.

An external feature, such as a shaft, is at MMC when the feature is at its maximum size limit. LMC is when the external feature is at its minimum size limit. An internal feature, such as a hole or slot, is at MMC when the feature is at its minimum size limit. LMC is when the internal feature is at its maximum size limit.

Rule #1, Perfect Form Boundary at MMC

The ASME standard uses the term "boundary of perfect form." The term *perfect form boundary* when used in this textbook has the same meaning. The most fundamental of the guidelines given in the dimensioning and tolerancing standard is



Goodheart-Willcox Publisher

Figure 5-6. Maximum material condition (MMC) is when the most material exists in the part. It is the largest shaft and the smallest hole that is permissible.

Rule #1. It defines what is often referred to as the *envelope principle*. The standard requires the surface or surfaces of a regular feature of size be within a boundary (envelope) of perfect form at MMC.

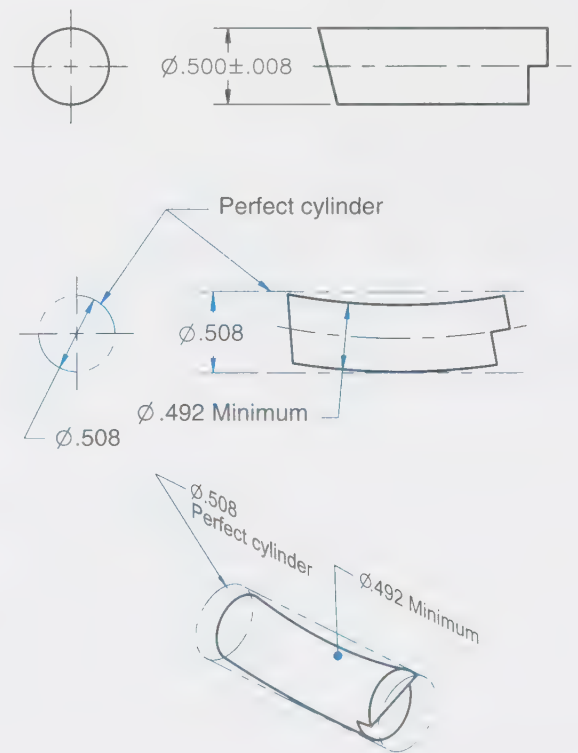
What Rule #1 means is this: When a feature is produced at MMC, that feature must have perfect form. See **Figure 5-7**. MMC size establishes a perfect form boundary and no part of the feature may extend outside the MMC boundary.

The example in **Figure 5-7** shows the effect of Rule #1 on a shaft, which is a cylindrical feature of size. The diameter dimension has an MMC value of .508". A theoretically perfect .508" diameter cylinder is the perfect form boundary for this feature.

As the size of a feature departs from MMC, its form is permitted to vary. Any amount of variation is permitted, provided the perfect form boundary is not violated. Of course, the size limits must not be violated.

It is possible for the given cylinder to be produced with form variations equaling .016" if the produced diameter of the part is only .492". It is not likely that this extreme condition will exist, but it is permissible according to the requirements of Rule #1.

From the given figure, it can be seen that perfect form of the feature is not required when



Goodheart-Willcox Publisher

Figure 5-7. Perfect form is required at MMC. The part may vary in form as it departs from the MMC size.

the feature is at LMC. *Perfect form at the LMC is not required even when all cross-sectional measurements (actual local size) along the length of the part are at LMC.* This fact is probably overlooked or misunderstood as much as any other item related to dimension interpretation. Although a requirement for perfect form at LMC can be established through specific tolerancing methods, it is important to know that a size tolerance does not by itself establish a requirement for perfect form at LMC.

The interrelationship between features is not affected by the perfect form boundary requirement of Rule #1. See Figure 5-8. When a regular feature of size is at MMC, it must have perfect form. This does not require that it have perfect orientation to any other feature.

When each of the three features of size in Figure 5-8 is at MMC, there remains an allowable tolerance on the 90° angle between the features. In this example, the given angularity tolerance of ±0.5° is permitted on each angle.

Rule #1 establishes a perfect form boundary at MMC for individual features of size. It does not establish any control of interrelationship between features of size.

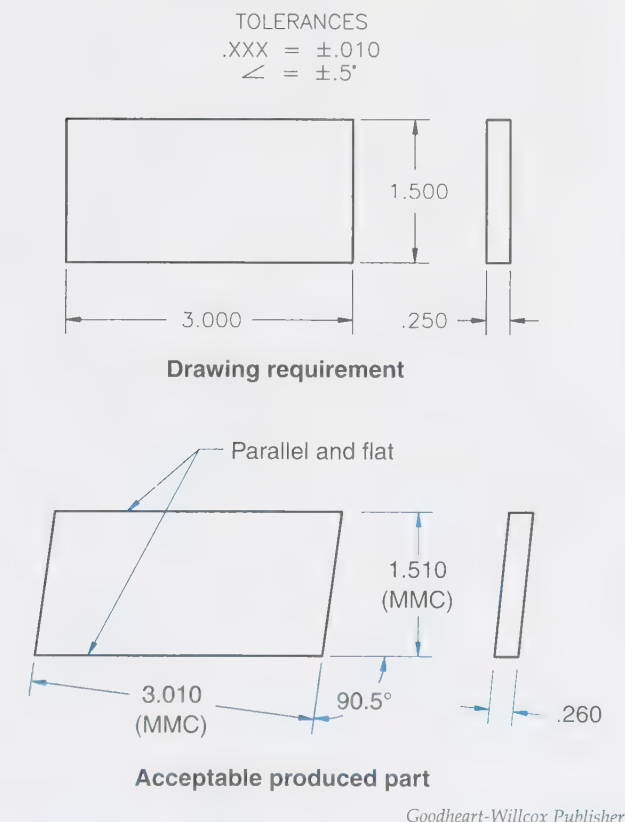


Figure 5-8. Perfect form at MMC is only a requirement on individual features. Features at MMC do not have a required perfect orientation to other features.

Perfect Form Boundary Effect on Flat Surfaces

A size dimension, such as thickness, applied to the distance separating two flat surfaces must have a size tolerance. See Figure 5-9. The distance measured between the two surfaces at any location must be within the acceptable limits of size. There is no requirement for all measurements to be of a single value.

If the limits of size for a feature are known, it is possible to determine the amount of form variation that can occur on the feature. The given figure shows a size dimension of 1.000" ±.010" applied to an external feature bounded by two parallel flat surfaces. The feature has size limits of .990" and 1.010"; therefore, a perfect form boundary made of two parallel planes exists at the 1.010" limit.

If the feature is produced at MMC (1.010"), then the two surfaces coincide with the planes of the boundary and must be perfectly flat and parallel. As the actual produced size varies from MMC, the surface form and orientation may vary, provided the perfect form envelope is not violated. This means that the acceptable surface variations are equal to the amount of size departure from the MMC value. The maximum possible form variation is equal to the size tolerance. In the given figure, the permitted form variations on one surface could be as much as .020".

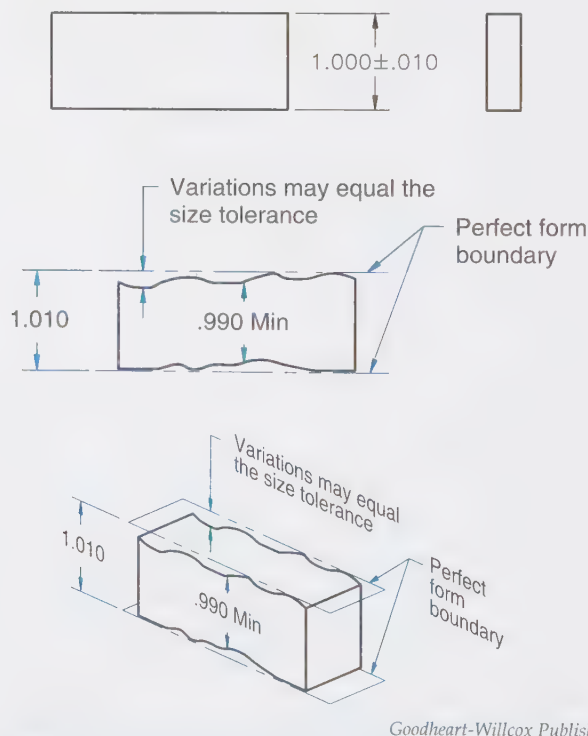


Figure 5-9. Form variation within the perfect form boundary may equal the size tolerance.

The acceptable amount of form variation on a feature is dependent on the produced (measured) size of the feature. The permitted form variation is equal to the amount of size departure from the MMC size. Size departure is the difference between the actual local size and the MMC value.

Perfect Form Boundary Effect on Cylinders

The perfect form boundary established by a diameter dimension is a cylinder. The perfect form boundary for a shaft is a cylinder equal to the maximum size limit of the shaft. The perfect form boundary for a hole is a cylinder equal in diameter to the minimum size limit of the hole.

Figure 5-10 shows a shaft with a $1.000'' \pm .010''$ diameter dimension. This results in a perfect form cylinder of $1.010''$ diameter. At any point along the length of the cylinder, the measured diameter (actual local size) may be any size between $.990''$ and $1.010''$. At any point where the diameter is less than $1.010''$, form variations on the shaft are permitted to equal the amount of departure from MMC. If the shaft is produced so that all cross-sectional measurements are $.990''$ (LMC), the shaft could have surface variations equal to $.020''$.

When form variations equal to the size tolerance are not acceptable, form tolerances or other geometric tolerances may be applied to the drawing.

Exceptions to Rule #1

The requirements of Rule #1 do not apply to everything. Exceptions include parts that are subject

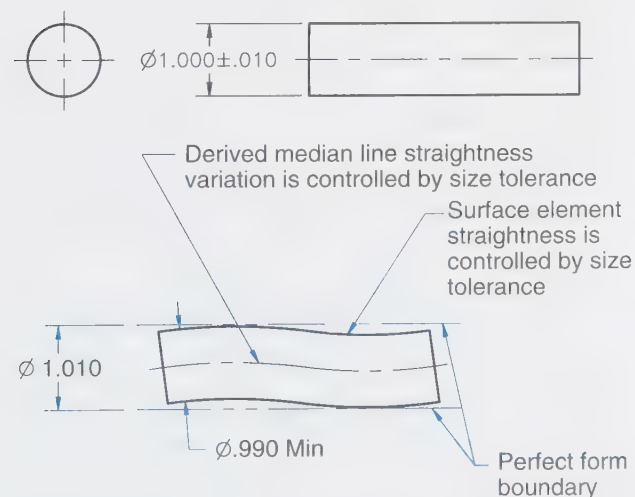


Figure 5-10. Both the surface variation and derived median line straightness variations of a cylinder are controlled by the diameter dimension.

to free state variation. Some stock materials are also excluded from Rule #1.

Free state variation is not well defined by the ASME Y14.5 standard. Some judgment must be used to determine when a part is subject to the extent of free state variation that is intended by the standard. The following examples of free state variation are given to provide guidance. A thin metal gasket will have a small size (thickness) tolerance. It is not practical to expect the part produced from thin stock to be within a perfect form envelope at MMC because the thin material deflects so easily. Another example is a shaft with a long length-to-diameter ratio. It may tend to flex, and this type of free state variation makes compliance with the perfect form envelope impractical.

A thin-walled tube when machined in a lathe chuck may meet the perfect form envelope requirement while clamped in the lathe chuck. Because the part is produced while under the forces exerted by the chuck and the stresses in the stock material, the tube will change shape when it is released. This is a type of free state variation.

Parts subject to free state variation may need to be measured prior to removal from machining clamps. It is also possible to use drawing notes that require the part to be restrained for inspection. Circular parts subject to free state variation are sometimes inspected by making multiple measurements to establish an average diameter.

Some stock material shapes have standardized form tolerances that are larger than the size tolerances for the shapes. Generally, Rule #1 does not apply to standard stock shapes. Sheet, plate, rod, and bar stock all have form tolerances that are larger than the required size tolerance. Extrusions and structural shapes also have form tolerances that exceed size tolerances.

It is also possible to specifically indicate that Rule #1 is not applicable to a size dimension. How to take exception to Rule #1 will be explained later in this chapter.

Straightness

A **straightness** tolerance may be applied to define the allowable straightness variation of surface elements or of a derived median line. The straightness tolerance specifies how close to perfectly straight a feature must be made. The allowable form variation for any surface with straight line elements may be controlled with this type tolerance. The form of the derived median line for a regular feature of size, such as a shaft, may also be controlled with a straightness tolerance.

A complete and accurate explanation of straightness tolerance requires precise terminology. Most of the terminology is defined in the current national standard. A **surface line element** is a line on the surface of a part. It may be referred to as a line element or a surface element. A **derived median line** is a line that is either straight (on a perfect part) or imperfect (on a part with surface variations) and is made of midpoints between surface points along two line elements on opposite sides of a feature. The derived median line is commonly called a centerline or an axis, but neither of these common terms is identical in meaning. A **center line**, for the purposes of tolerances, is a line that represents the center of a feature on a drawing or in a CAD model. An **axis** is always an exactly straight line about which a feature or part is rotated. The standard currently defines an **axis** as a line at the center of an **actual mating envelope** for the feature. So, a center line and an axis are by definition straight lines and have no variation in them. It is the derived median line that varies based on the surface condition of a produced shaft or hole.

The meaning of a straightness tolerance specification varies according to the type of feature to which the tolerance is applied. How the straightness tolerance is shown on the orthographic views or in the CAD model indicates whether the tolerance is applicable to surface elements or a derived median line. In orthographic views, application of the straightness feature control frame on an extension line or on a leader pointing to a surface provides a specification that applies to a surface. Placement adjacent to a dimension or attached to a dimension line indicates the tolerance applies to the feature of size. A straightness tolerance applied to a feature of size has a completely different meaning than when applied to a surface.

Each straightness tolerance specification is shown in a feature control frame. See **Figure 5-11**. The feature control frame includes the straightness tolerance symbol and the tolerance value. The symbol indicates the required tolerance zone shape, and the tolerance value shows the allowable size of the tolerance zone. A datum feature reference is never shown in a straightness tolerance specification for the purpose of constraining the translational or rotational degrees of freedom for the tolerated feature.

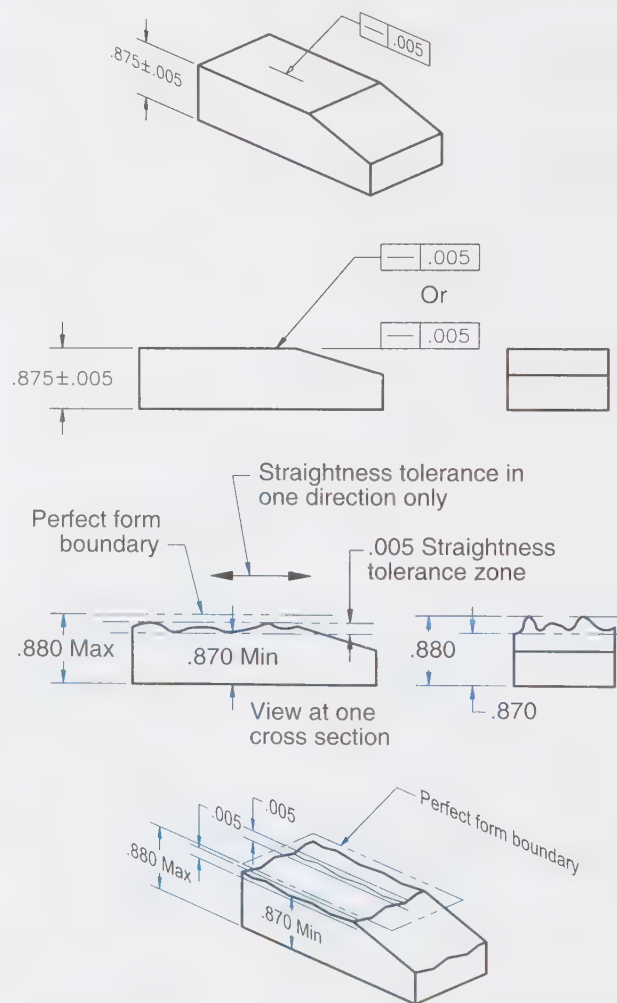
Applied to Surface Elements

Previous paragraphs in this chapter explained that form variations on a surface must be contained within the size tolerance. It is necessary to apply a

feature control frame containing a form tolerance when the form variations must be controlled to a smaller value than is required by the size tolerance. See **Figure 5-11**.

The given part has a size tolerance of $\pm .005$ " between the top and bottom surfaces. The maximum allowable form variation on the top surface based on this tolerance is as follows. If the bottom surface is perfect, the maximum form variations on the top surface are equal to the size tolerance of .010". If .010" variations on the top surface of the given part are acceptable, then no other specification is needed.

A form tolerance feature control frame is only applied to a surface when the required control is smaller than the size tolerance. The shown form tolerance is .005", but any value less than .010" could be applied without creating a conflict with the size tolerance. Application of a .010" form tolerance would not be wrong, but it would be



Goodheart-Willcox Publisher

Figure 5-11. Straightness tolerances on a flat surface define allowable variation for line elements in one direction.

redundant because the size tolerance already controls the surface to within .010".

The specified .005" straightness tolerance in the given figure is the maximum allowed straightness variation. That full .005" cannot be used when a part is produced at or near the MMC limit of size, because the part size and form variations must comply with Rule #1.

A part produced at MMC cannot use any of the specified straightness tolerance. A part that departed .001" from the MMC size is only permitted .001" form variation. Any form variation greater than .001" would result in violation of the perfect form boundary, and that is not allowed. A part produced .004" smaller than MMC is only permitted .004" form variation. Again, the size limits have controlled the allowable amount of form variation. Only when the size departs more than .005" from MMC does the specified form tolerance begin to have an effect.

If a part varied in size .007" from the perfect form boundary, the size tolerance would allow .007" form variation, except the specified straightness tolerance of .005" must be met. So in this case the straightness tolerance limits the allowable form variation when the size has departed from MMC by more than .005". The size tolerance and Rule #1 must be met, and the straightness tolerance must also be met. The table below shows the combined effects of the size and form tolerances based on the part in **Figure 5-11**.

It is important to notice that the straightness tolerance specification is unchanged by the change in feature size. The straightness tolerance remains .005 for all sizes of the feature. However, the allowable form variation is controlled by the size limits when the departure from MMC is less than the specified tolerance value. A part that is produced with a diameter of .878" and has a straightness variation of .005" is a bad part even though it has met the specified straightness tolerance. It is bad because it has failed to meet the requirement of Rule #1 that requires the feature to be within the perfect form boundary. The perfect form boundary requirement only allows a form variation of .002" when the feature is at the .878" size. This effect of Rule #1 should not be confused with the allowable increase in specified tolerance that results from application of the MMC modifier on tolerances applied to features of size.

Because of Rule #1, a form tolerance larger than the size tolerance is not normally specified on a surface. Showing a form tolerance larger than the size tolerance creates a conflict between the Rule #1 requirements and the form tolerance. So, a form tolerance larger than the size tolerance is not permitted on a surface unless exception is taken regarding Rule #1. Exception is taken by placing the independency symbol adjacent to the size dimension value, as explained later in this chapter. The independency symbol indicates that the default requirement for perfect form at MMC is not applicable.

	Produced Size	Departure from MMC	Specified Straightness	Allowed Form Variation
MMC	.880	.000	.005	.000
	.879	.001	.005	.001
	.878	.002	.005	.002
	.877	.003	.005	.003
	.876	.004	.005	.004
	.875	.005	.005	.005
	.874	.006	.005	.005
	.873	.007	.005	.005
	.872	.008	.005	.005
	.871	.009	.005	.005
LMC	.870	.010	.005	.005

Material condition modifiers are not used in form tolerance specifications when they are applied to surfaces.

Straightness of Flat Surfaces

The straightness requirements for line elements on a flat surface are specified as shown in [Figure 5-11](#). The feature control frame is drawn to show the straightness symbol and the allowable tolerance value. The feature control frame when shown in an orthographic view is applied to the surface by a leader, or it may be placed on an extension line from the surface. In a CAD model, a line is drawn on the surface to indicate the direction of the straightness tolerance, and the leader from the feature control frame terminates in an arrow that contacts the line.

It is important to select the correct orthographic view for application of a straightness specification. Straightness tolerance specifications only define the required straightness of line elements that are parallel to the line on which the specification is shown. The feature control frame in a CAD model is typically attached by a leader and it points to a line on the surface. The line on the surface indicates the direction of the tolerated line elements. Straightness applied to a flat surface should not be confused with flatness.

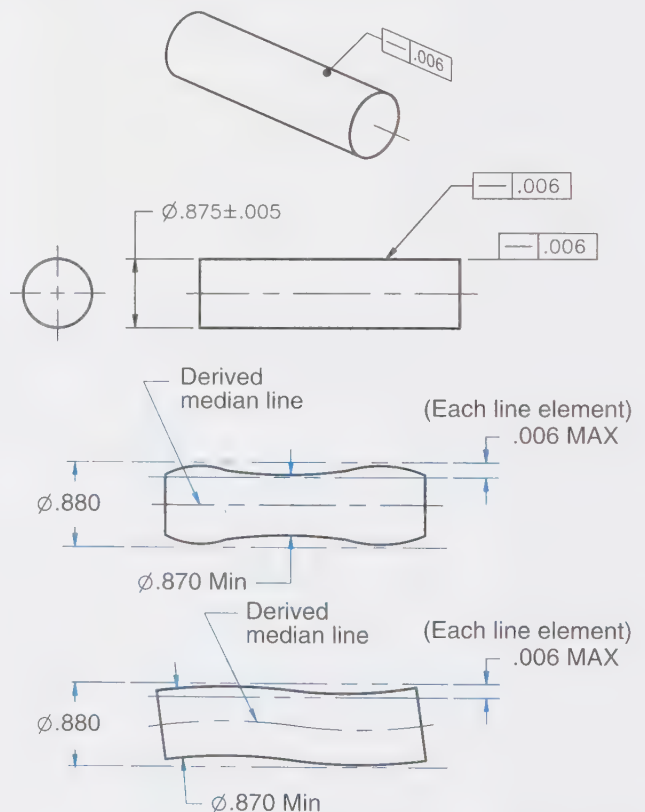
[Figure 5-11](#) shows the interpretation of a size tolerance and a straightness tolerance. The size dimension between the top and bottom surfaces has a maximum limit of .880". The part has a perfect form boundary at the dimension of .880", and no surface variations may extend outside this boundary. The size dimension has a minimum limit of .870", and no measurement across the part may be less than this value.

The straightness tolerance in the given drawing is applied to the top line of the front view; therefore, all surface elements parallel to this line must be straight within the given .005" tolerance. In a CAD model, the tolerance is connected to the feature by a leader that points to a line on the top surface and all surface elements parallel to this line must be straight within the specified .005". Each line element on the produced part is verified separate from all others because the specification is straightness, not flatness. Line elements perpendicular to the required direction of control may be out-of-straight by an amount equal to the size tolerance. In the given figure, the straightness tolerance applied to the top surface does not imply any requirements for parallelism to the bottom surface.

Straightness of Cylinder Surface Elements

Two types of straightness tolerance may be specified on a cylinder. One defines the allowable variation on surface element straightness. The other defines the allowable variations of the derived median line. Straightness tolerances applied to straight surface elements on a cylinder are explained in this section. Straightness tolerance to specify allowable derived median line variation is achieved in a different manner than surface straightness and is explained later in this chapter.

A surface element straightness tolerance is indicated by attaching the feature control frame to the surface of a cylinder by either a leader or an extension line. See [Figure 5-12](#). The same tolerance specification in a CAD model is achieved with a leader attached to the surface of the cylinder and terminating the leader with a dot instead of an arrow. Proper application of a straightness feature control frame is very important when applying the tolerance to a cylinder. Incorrect application will change the meaning of the specified tolerance.



Goodheart-Willcox Publisher

Figure 5-12. Straightness tolerances on a cylindrical surface only control line elements on the cylinder.

Allowable surface variations on a cylinder are established by the size tolerance just as they are for a flat surface. As previously stated, Rule #1 requires a perfect form boundary at the maximum material size. Application of a form tolerance is only necessary when the maximum desired form variation is a smaller value than the size tolerance.

The given example shows a shaft with a dimensioned diameter of $.875" \pm .005"$. This results in a size tolerance of $.010"$. A straightness tolerance of $.006"$ is applied to the surface. Rule #1 applies and there is a perfect form boundary at $.880"$ diameter.

Two of the many possible part configurations that could result from the given size and straightness specifications are shown. In the first example, the $.880"$ diameter perfect form boundary is shown. None of the part variations may fall outside this boundary. The smallest diameter at any point along the shaft must not be less than the smallest size limit, $.870"$. The variations of surface straightness must also be within the specified $.006"$ straightness tolerance.

Surface variations along the length of the part may be caused by a change in diameter (the actual local size). The change in diameter causes surface straightness variations, but on the first of the shown parts, the derived median line has remained perfectly straight. This example shows that a straightness variation on the surface of a cylinder may not affect the derived median line straightness. It also shows that straightness of a derived median line may not control the surface.

The second example shows the same $.880"$ diameter perfect form boundary. The smallest allowed measured diameter is $.870"$. Each surface element on the shaft is straight within the allowable $.006"$ tolerance. On this part, the surface variations are such that the shaft derived median line is not straight.

A straightness tolerance applied to the surface of a cylinder controls the surface elements. It does not control the derived median line (axis) straightness. The straightness variations on each element are independent of variations on any other element. Regardless of the specified surface form tolerance, the derived median line straightness variations may be as much as is permitted by the size tolerance. There is no orientation requirement included in the straightness tolerance.

The straightness tolerance on the surface of the part establishes a requirement that must be met. The size tolerance establishes another requirement that must be met and includes a perfect form boundary at MMC. If the shaft in [Figure 5-12](#) is

produced at a diameter of $.880"$, it is at the MMC envelope and must have perfect form. At $.880"$, the $.006"$ straightness cannot be used or the surface will go outside the perfect form boundary.

If the shaft measures $.879"$ at all locations, it has departed from the MMC envelope by $.001"$. It cannot use the full $.006"$ straightness tolerance because there is only $.001"$ departure from the perfect form boundary. Any form variation greater than $.001"$ will go outside the perfect form boundary and is not acceptable. So, a part produced at $.879"$ with a straightness variation of $.006"$ is not acceptable. The specified straightness tolerance of $.006"$ is not violated, but the perfect form boundary at MMC is violated. The part would be rejected because of violation of the size limits (outside the perfect form boundary).

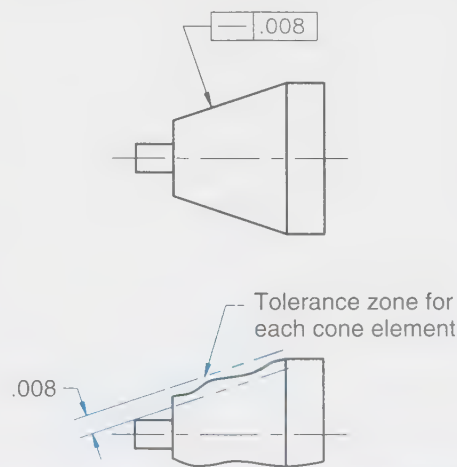
Another way to think of this is that the straightness tolerance cannot all be used unless the size has sufficiently departed from the perfect form boundary.

Straightness of Cone Elements

Straightness tolerances may be applied to any feature that has straight line elements. See [Figure 5-13](#). A cone is an example. Straightness of the elements that lie in a plane with the cone axis may be controlled. Each cone element is controlled independently. Although the straightness tolerance zone must be in a common plane with the cone axis, there is no requirement to maintain a specific angle to the axis. The cone angle is controlled either by the size and angle tolerances, or by a profile tolerance, which is defined in another chapter.

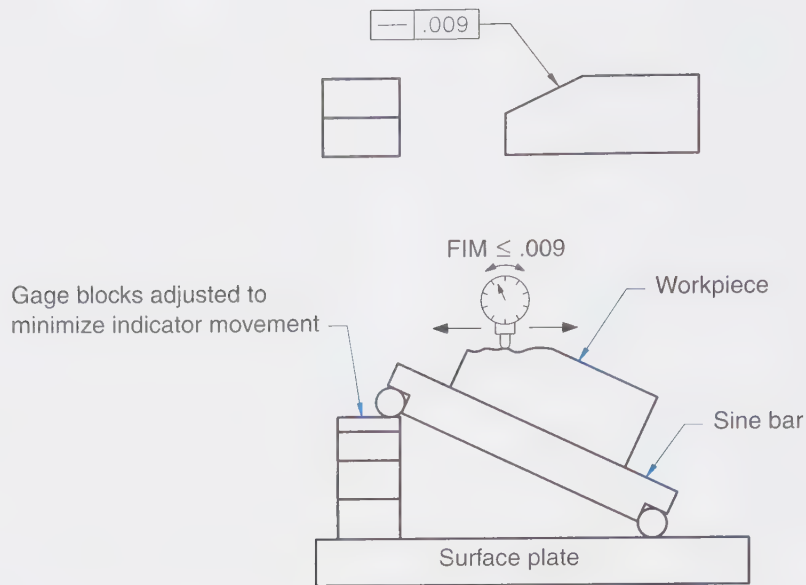
Verification of Surface Straightness

The part in [Figure 5-14](#) has a surface straightness requirement of $.009"$ on the inclined surface.



Goodheart-Willcox Publisher

Figure 5-13. A straightness tolerance may be applied to any feature that contains straight line elements.



Goodheart-Willcox Publisher

Figure 5-14. Straightness only controls form. The orientation of a part may be adjusted to minimize measured variations because the form tolerance does not control orientation or location.

The surface elements parallel to the edge of the part must be straight within the specified .009" tolerance. There is no other requirement implied by the straightness tolerance. Surface straightness inspection is completed independent of the size requirement.

One method for inspecting the straightness of the surface elements is shown. The part is set on an inclined *sine bar*. The sine bar is inclined on top of a flat plate or surface table by placing gage blocks under one of its feet. Measurement of straightness variations should not require a specific orientation of the part during measurement; therefore, the angle of the sine bar may be adjusted to minimize the dial indicator readings.

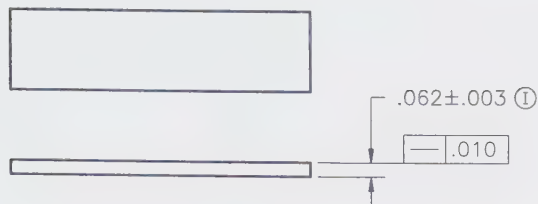
A dial indicator is moved in a straight line along the surface. This is repeated at several locations across the surface. The variations measured on each pass of the dial indicator are considered separately. The variation on any one pass is determined by the full movement of the dial indicator needle. This is referred to as the *full indicator movement (FIM)* or *total indicator reading (TIR)*.

Another method to verify acceptability would be to place a straightedge along the surface and use a .009" feeler gage to check for excessive gaps between the surface and the straightedge. If surface irregularities are such that the feeler gage blade width is a problem, a .009" wire gage may be used. In the case of a convex surface, care must be used because this method can result in measurement values greater than the actual straightness variation.

Noted Exceptions to Rule #1

The requirement for perfect form at MMC can be very restrictive when the size tolerance is small. At times, this requirement can make production of the part expensive, if not impossible. If the size limits for a part make production within the perfect form boundary difficult, a decision must be made as to whether or not an exception to Rule #1 will be taken. The requirement for perfect form at MMC must be considered when designing a part, and the functional design should be the final determining factor in whether or not to take exception to Rule #1.

Variance outside the perfect form boundary may need to be permitted for a large, thin part produced with a small thickness tolerance. A part may be permitted to vary outside the perfect form boundary by specifying a form tolerance that is larger than the size tolerance and also applying the independency symbol to the size dimension. The independency symbol indicates Rule #1 is not applicable to a specific dimension. See [Figure 5-15](#). The independency symbol is applied to the thickness dimension to indicate that Rule #1 is not applicable. With the independency symbol applied, the size tolerance only controls thickness and has no effect on form. A form tolerance is therefore applied to one surface to limit the allowable form variation. Before the independency symbol was standardized in 2009, the practice was to include a note adjacent to the size dimension to indicate PERFECT FORM AT MMC NOT REQUIRED.



Goodheart-Willcox Publisher

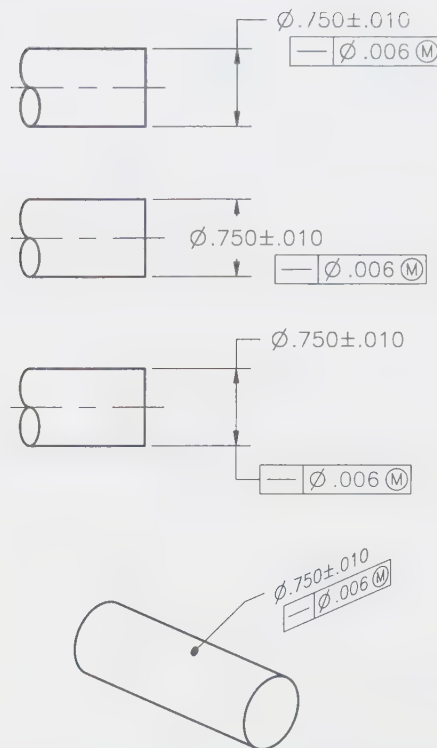
Figure 5-15. If a form tolerance larger than the size tolerance is applied to a surface, an independency symbol must be applied to the size dimension to take exception to the requirements of Rule #1.

Exception to Rule #1 should only be taken when required for production and when the functional design is such that the specified variation is acceptable. When taking exception to Rule #1, it is necessary to show both the independency symbol and a geometric tolerance to control form. Application of a form tolerance larger than the size tolerance is an error if the independency symbol is not present. Application of the independency symbol without a geometric tolerance to control form would also be an error. Omission of the form control when exception to Rule #1 is specified would result in an unknown amount of acceptable form variation. Therefore, a tolerance specification that controls form is essential if exception to Rule #1 is specified.

Applied to Features of Size

Straightness requirements of a derived median line for a cylinder may be specified through a form tolerance applied to the regular feature of size. This form tolerance application is sometimes referred to as axis straightness, but that is an inaccurate term. An axis is by definition a straight line, so straightness variation cannot exist for an axis. The tolerance is applied to a cylindrical feature of size by placing a feature control frame on the diameter of a shaft or hole. See Figure 5-16. The feature control frame may be placed adjacent to the size dimension, or it may be attached to the dimension line. All of the methods shown in Figure 5-16 indicate that the derived median line is being controlled. When a form tolerance is applied to a regular feature of size, the requirements of Rule #1 are not applicable to that feature.

The method used in a CAD model is similar to one of the methods used for surface element



Goodheart-Willcox Publisher

Figure 5-16. The derived median line of a feature may be controlled by placing a form tolerance feature control frame adjacent to the size dimension or on the dimension line.

straightness, but there are differences that indicate the tolerance is applicable to the derived median line and not to the surface elements. In a CAD model, the feature control frame is placed adjacent to the size dimension. The leader connects the size dimension and feature control frame to the cylindrical surface and is terminated with a dot. The placement near the size dimension and the presence of a diameter symbol in the feature control frame are the indicators that the tolerance applies to the derived median line and not to the surface elements.

In orthographic views, to establish a tolerance on the derived median line, care must be used to avoid making contact between the feature control frame and any extension lines from the surfaces. If an extension line is contacted, it will appear the intent is to establish surface straightness instead of derived median line straightness.

Straightness of the derived median line is not restricted by the limits of size (Rule #1) when a straightness tolerance is specified on the feature of size. In fact, Rule #1 does not apply when straightness tolerance is applied to a feature of size.

It is permissible to apply a straightness tolerance that is larger than the size tolerance on a feature of

size. This does not require an exception to Rule #1 be noted because a straightness tolerance applied to a feature of size overrides the requirements of Rule #1.

A straightness tolerance applied to a feature of size is assumed to apply RFS unless otherwise shown. Each of the given examples includes the MMC modifier. When the tolerance specification is applied to control straightness of the derived median line for a cylinder, a diameter symbol is placed in front of the tolerance value to indicate that the tolerance zone is a cylinder.

Straightness to Control a Shaft Derived Median Line

A derived median line straightness specification applied to a shaft creates a cylindrical tolerance zone that extends the full length of the shaft. See **Figure 5-17**. The figure shows a size and straightness tolerance applied in a partial view, and the lower portion of the figure shows the effect of the size tolerance and straightness tolerance. The straightness tolerance is specified to apply regardless of feature size (RFS). There is a .006" diameter cylindrical zone that extends the length of the shaft. This tolerance zone does not establish a tolerance zone for the surface elements, but instead defines the allowable form tolerance applicable to the derived median line. The variations in the surface of the shaft may be of any value

permitted by the size tolerance, provided the effect on the derived median line does not cause any violation of the .006" diameter zone.

Diameter measurements made at any cross section (the actual local size) along the shaft must be within the limits of size—between .740" and .760" diameter. The straightness tolerance specification permits a .006" diameter derived median line straightness variation regardless of the feature size. For this particular part, whether all cross-sectional measurements are .760" or vary between .740" and .760", the derived median line straightness variation is permitted to be .006".

The given figure shows the effect of a .006" straightness variation with the shaft produced at its maximum size limit of .760". A derived median line straightness variation of .006" with the shaft at .760" creates a maximum outer diameter of .766".

Virtual Condition of a Shaft

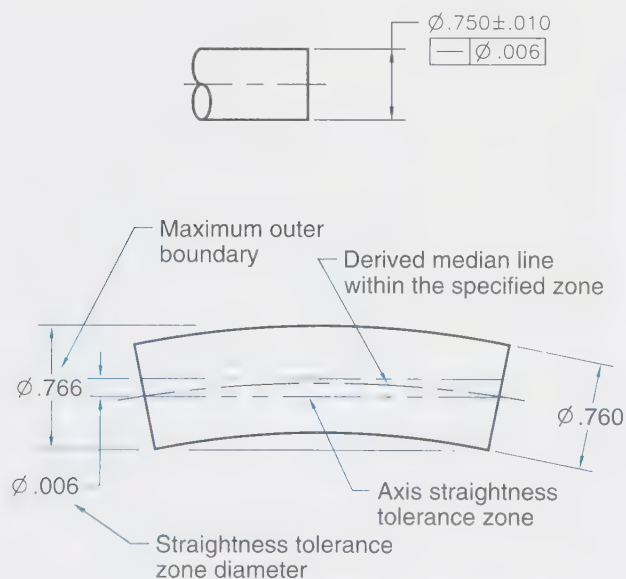
A geometric tolerance applied with the MMC modifier creates a boundary known as a **virtual condition**. See **Figure 5-18**. The virtual condition is the MMC size and the effects imposed by the applied tolerances. Another way to think of the virtual condition is to visualize the actual mating envelope required to contain the combined effects of the MMC size and the tolerance zone. The shape of the actual mating envelope is a theoretically perfect geometric counterpart. For a hole or shaft, the envelope is a perfect cylinder.

The virtual condition for a shaft can be determined by adding the MMC size of the shaft and the derived median line straightness tolerance. See **Figure 5-18**. The virtual condition for the given shaft is 1.267". This was determined by adding the straightness tolerance of .007" to the MMC size of 1.260".

The concept of a virtual condition is important to understand. Calculations for parts that must slide together are typically completed based on virtual conditions. If the virtual conditions of two parts in an assembly will fit together, then any part that is within the virtual condition envelope will also fit in the assembly. This is a primary justification for using MMC modifiers on tolerance specifications.

An MMC modifier applied to a derived median line straightness tolerance permits the tolerance zone to increase as the feature size departs from MMC. Although the straightness tolerance is being allowed to increase, the virtual condition envelope that encloses the shaft does not change.

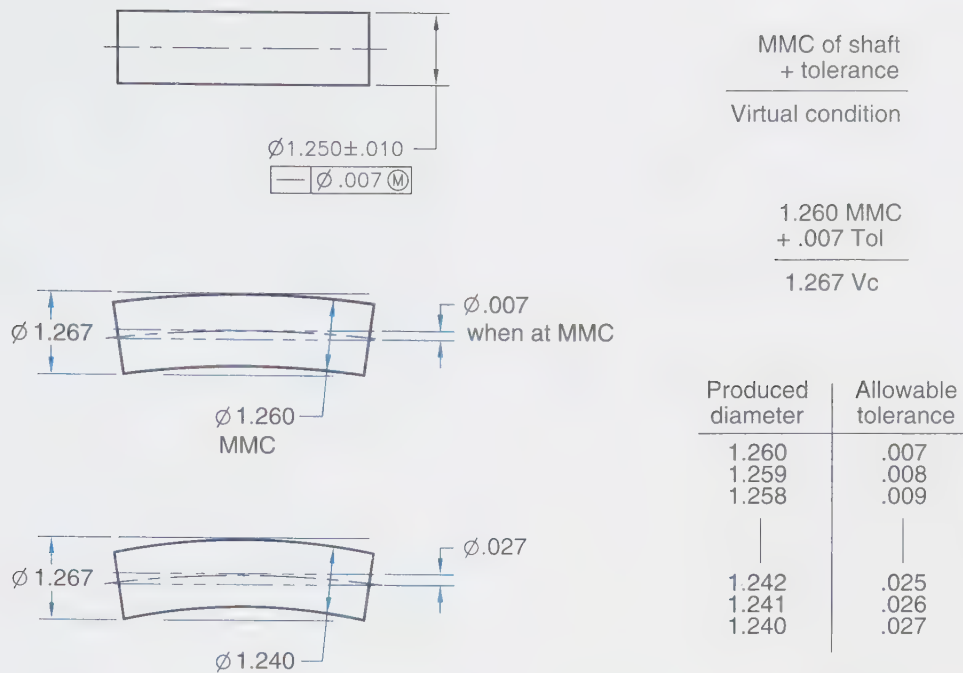
A part produced at MMC has an allowable form tolerance equal to the value shown in the feature



No perfect form boundary exists at MMC

Goodheart-Willcox Publisher

Figure 5-17. A straightness tolerance applied to a feature of size permits the part to have derived median line straightness variation when the part is at MMC.



Goodheart-Willcox Publisher

Figure 5-18. The permitted derived median line straightness variation on a cylinder has an effect on the apparent diameter of the cylinder. The combined effect of the MMC size and the permitted straightness variation is known as the virtual condition.

control frame. For the given figure, a 1.260" MMC diameter shaft is allowed a straightness tolerance of .007". This results in a virtual condition of 1.267" diameter.

The tolerance of .007" diameter shown in the given figure is only required while the shaft is at MMC (1.260"). As the diameter of the shaft gets smaller, the straightness tolerance is permitted to increase. The tolerance zone increases by an amount equal to the departure of the shaft from its MMC size.

If a part is produced with a diameter smaller than 1.260", that part can have a larger straightness variation and still fit inside the same virtual condition envelope. The amount of departure from MMC is the amount of additional tolerance that can be added to the value shown in the feature control frame.

The given table shows various produced shaft diameters and the allowable straightness tolerances. Each diameter and corresponding tolerance, when added together, equals the virtual condition. The increased straightness tolerance is not creating a worse case than the virtual condition created by the specified tolerance at MMC.

Because the straightness tolerance increases by an amount equal to the departure from MMC, the example shaft is permitted a straightness variation of .027" when the shaft diameter is 1.240".

Increasing the straightness tolerance zone as the shaft size decreases results in an envelope that remains the same size as the virtual condition.

An MMC modifier should be used when the function of the part only requires clearance for assembly. If the derived median line straightness tolerance must remain unchanged regardless of the produced size, then the MMC modifier must be omitted, and the default RFS used. The impact of the RFS application is important to understand. If the tolerance in the given figure is specified at RFS, then .007" maximum variation is required regardless of the produced diameter of the shaft.

Bonus Tolerance (Additional Tolerance)

The maximum allowable amount of straightness variation can easily be calculated for any produced diameter when the MMC modifier is used. The allowable straightness tolerance is determined by adding the tolerance in the feature control frame to what is commonly referred to as the bonus tolerance. The **bonus tolerance** for a particular part is determined by finding the difference between the produced size and the specified MMC size. Note: The term *bonus tolerance* is not in the ASME standard but is used here based on its common understanding in industry.

The following example shows how calculations are made to determine the allowable tolerance

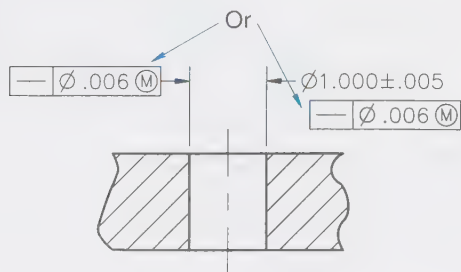
for a produced part. The example is based on the specified size and form tolerance in [Figure 5-18](#).

Specified MMC	1.260
Produced shaft diameter	<u>-1.248</u>
Bonus tolerance	.012
Specified form tolerance	.007
Bonus tolerance	<u>+ .012</u>
Total allowable form tolerance	.019

Straightness of the Derived Median Line of a Hole

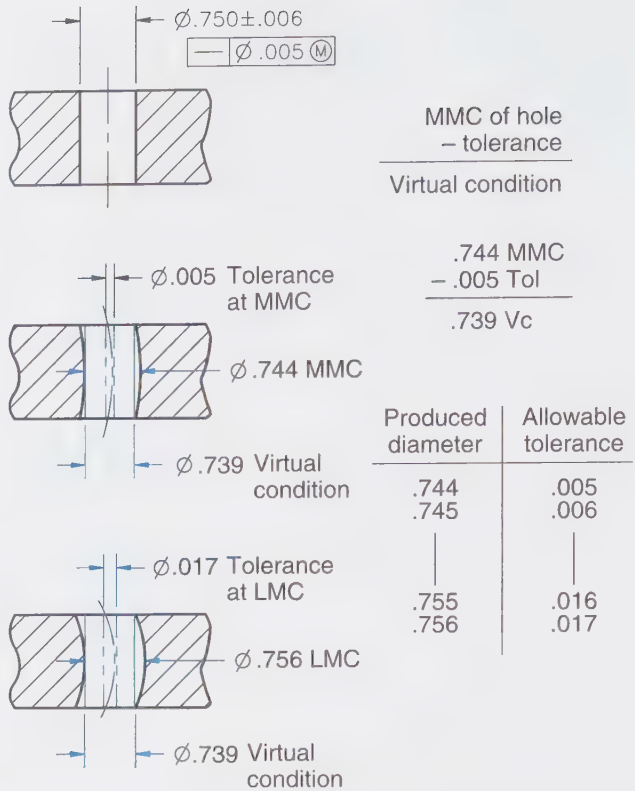
A straightness tolerance defining allowable variation on the derived median line of a hole may be applied in either of two ways. See [Figure 5-19](#). In orthographic views, the feature control frame may be applied to the dimension line for the hole diameter, or it may be placed adjacent to the diameter dimension value. In a CAD model, the feature control frame is placed adjacent to the size dimension. The feature control frame must not be placed on an extension line if the intent is to define a tolerance for the derived median line. Placement on an extension line would result in control of the surface line elements rather than the derived median line.

A derived median line straightness specification applied to a hole creates a cylindrical tolerance zone that extends the full length of the hole. When applied with the MMC modifier, it creates a virtual condition that extends the full length. See [Figure 5-20](#). The figure shows the combined effect of the size tolerance and straightness tolerance at MMC. The straightness tolerance creates a .005" diameter cylindrical zone that extends the length of the hole for cases where the size of the hole is at MMC. If the hole is produced at any size other than MMC, then the size of the tolerance zone is changed based on the allowable bonus tolerance. If the diameter of a single hole varies along its length, compliance with the tolerance zone may become difficult to measure. It may be easier to verify that the surface of the hole has not violated the virtual condition.



Goodheart-Willcox Publisher

Figure 5-19. A form tolerance may be applied to internal features such as holes.



Goodheart-Willcox Publisher

Figure 5-20. The virtual condition of a hole has a diameter smaller than the MMC diameter permitted by the size dimension.

Virtual Condition of a Hole

The combined effect of the MMC diameter of a hole and the derived median line straightness tolerance creates a condition known as the virtual condition. When a tolerance is specified with the MMC modifier, virtual condition is the MMC size and the effect of the applied tolerances. Another way to think of the virtual condition for a hole is to visualize the maximum actual mating envelope that fits inside the hole when the hole is at MMC and the straightness tolerance is applied.

The virtual condition for a hole can be determined by subtracting the form tolerance from the MMC size. See [Figure 5-20](#). The virtual condition for the given hole is .739".

An MMC modifier applied on a derived median line straightness tolerance permits the tolerance zone to increase as the actual local size of the feature departs from MMC. As the hole diameter increases the straightness tolerance is allowed to increase, but the mating envelope inside the hole does not violate the virtual condition boundary.

The tolerance of .005" diameter shown in the given figure is only required while the hole is at MMC (.744"). As the diameter of the hole gets larger, the straightness tolerance is permitted to

increase. The tolerance zone increases at a value equal to the departure of the hole from its MMC size.

Because the straightness tolerance increases by an amount equal to the departure from MMC, the example hole is permitted to have a straightness variation of .017" when the hole diameter is .756". Increasing the tolerance zone diameter by an amount equal to the increase in hole size results in an actual mating envelope that would not violate the virtual condition.

The tolerance should be specified at MMC when the function of the part only requires clearance for assembly. If the function of the part requires the derived median line straightness tolerance remains unchanged regardless of the produced hole size, then the MMC modifier must be omitted from the straightness tolerance and RFS assumed. If the straightness tolerance is specified at RFS, then the .005" straightness requirement applies regardless of the produced diameter of the hole.

Verification of Derived Median Line Straightness

Tolerances specified at MMC may be inspected using functional gages or measurement methods that can determine if features have complied with a required virtual condition. *Functional gages* generally have the benefit of making inspection faster and easier than using free-standing manual inspection methods. Design and fabrication costs for gages are recovered when the gages are used

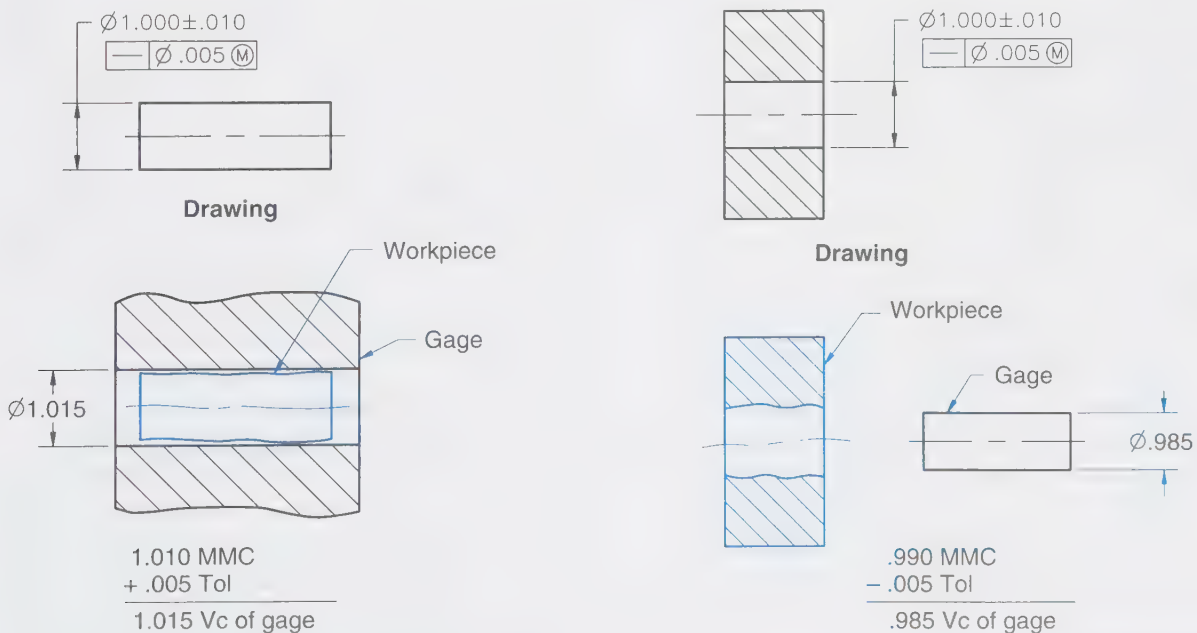
to inspect large numbers of parts. If production quantities are not large enough to justify the cost of gages, then free-standing inspection methods may be used.

A shaft with a derived median line straightness tolerance may be inspected with a functional gage that contains a hole equal in diameter to the virtual condition of the shaft. See [Figure 5-21](#). The theoretical gage hole diameter of 1.015" for the shown shaft is determined by adding the shaft MMC of 1.010" to the straightness tolerance of .005". This is the theoretical size requirement for the hole in the gage. Gage tolerances would be applied to this diameter to establish the gage fabrication requirements.

The functional gage in the figure only verifies the straightness tolerance. The size limits of .990" and 1.010" must be verified with other gages or diameter measurements made with a caliper, micrometer, or other instrument.

The functional gage used to check derived median line straightness of a hole is a perfect form shaft that has a theoretical diameter equal to the virtual condition of the hole. See [Figure 5-21](#). The shown gage pin diameter of .985" is determined by subtracting the .005" straightness tolerance from the .990" MMC of the hole. The .985" diameter is the theoretical size required for the gage pin.

No attempt has been made to explain how to assign gage tolerances to the theoretical gage dimensions. Dimension and tolerance calculation for gages is an extensive subject and is not required



Goodheart-Willcox Publisher

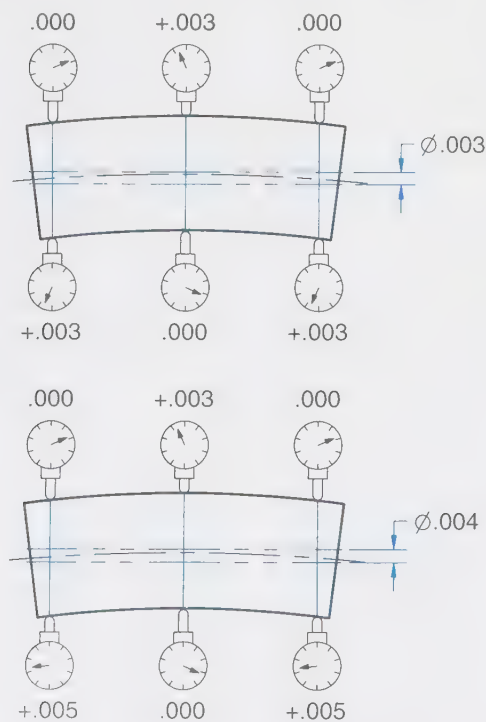
Figure 5-21. Functional gages sized at the virtual condition of a feature may be used for inspection if the form tolerance is specified with an MMC modifier.

to understand dimensioning and tolerancing of production parts. Standard guidelines for functional gage design are contained in ASME Y14.43.

The use of fixed size functional gages is generally limited to inspection of tolerances that are specified with the MMC modifier. If tolerances are specified to apply RFS, then parts may be verified using free-standing manual inspection methods or gages with adjustable features. See **Figure 5-22**. Computer-driven *coordinate measuring machines (CMM)* and other precision systems may be used to verify tolerances specified at MMC or RFS.

Verification of derived median line straightness with a free-standing setup may be accomplished by measuring surface variations on opposite sides of the cylinder. Care must be taken to observe locations at which measurements are being taken, because the axis variation is calculated from the surface measurements. In **Figure 5-22**, the surface variations are such that the derived median line variation is calculated to be one-half the total surface variation. The derived median line variation is not always half the surface variation as it is in **Figure 5-22**. Also, refer back to **Figure 5-12**.

It is necessary to take multiple sets of measurements to determine the full amount of straightness variation. Attempting to determine derived median line location from surface measurements



Goodheart-Willcox Publisher

Figure 5-22. Free-standing inspection methods may be used when functional gages are not available.

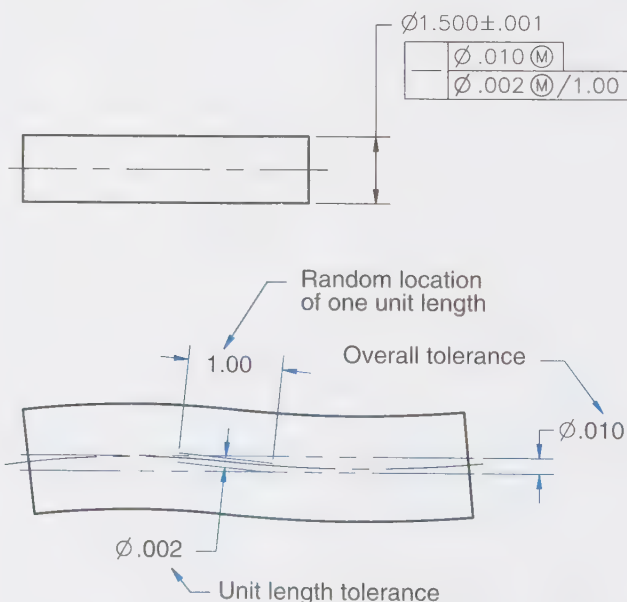
can be frustrating, and the accuracy of the calculated variation is subject to being questioned.

Overall and Unit Length Tolerance

Situations sometimes occur making it necessary to specify straightness of the derived median line within a small tolerance zone over a unit length of the cylinder. Care must be taken to make the tolerance zones no smaller than is needed, because small tolerance zones increase part cost. If taken to an extreme and the tolerance zone is excessively small, the parts cannot be produced.

Achievable tolerances are affected by the proportions of the part. A relatively small straightness tolerance can be achieved on a shaft that has a small length-to-diameter ratio. A relatively large straightness tolerance may be needed for the full length of a shaft that has a large length-to-diameter ratio.

Sometimes it is necessary to confine a segment of a long shaft to a small straightness tolerance. In these situations, the part can be made more producible by applying both an overall and unit length straightness tolerance. **Figure 5-23** shows a relatively large straightness tolerance applied to the full length of a cylinder. It is combined with a more restrictive tolerance that only applies to a unit length. This permits the tolerance zone that affects the overall length to be made as large as possible while specifying a unit length of the shaft that must meet a smaller straightness tolerance.



Goodheart-Willcox Publisher

Figure 5-23. Overall straightness and straightness per unit length may be combined in a two-segment feature control frame.

A combined specification of overall and unit length straightness is placed in one feature control frame that contains an upper and lower segment. See **Figure 5-23**. Only one straightness tolerance symbol is shown. The upper segment shows the overall straightness tolerance. The second segment (second line) of the feature control frame shows the straightness tolerance per unit of length. The tolerance value per unit of length will always be smaller than the overall tolerance.

The given example shows an overall straightness tolerance of .010" diameter at MMC. The entire length of the derived median line must fall within this .010" diameter tolerance. The unit length tolerance is .002" diameter at MMC and is applicable to every 1.00" segment of part length. This means that along any 1.00" of length on the shaft, it must be straight within .002" diameter at MMC. If MMC modifiers are not shown, then the tolerances would apply RFS.

The unit length may be any distance value across which the straightness tolerance needs to be applied. The given example shows 1.00". It could be 1.50" or any other dimension required by the function of the part.

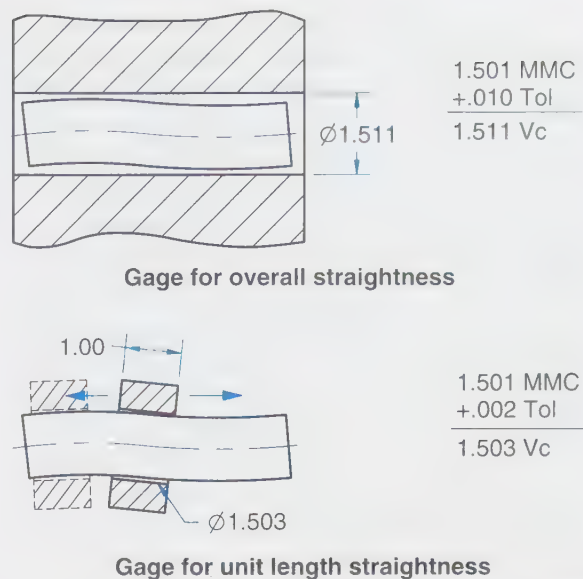
A straightness per unit length may be specified without any overall straightness tolerance, but the practice has few applications. Doing this would permit the unit straightness tolerance to accumulate along the length of the part. Design applications usually require that an overall tolerance be specified to control the accumulation of the unit length tolerance.

Verification of Unit Length Tolerance

A method for verifying overall straightness and unit length straightness tolerance requirements is shown in **Figure 5-24**. The given example is based on the dimensions in **Figure 5-23**. The gages and calculations shown in **Figure 5-24** are only applicable to tolerances specified at MMC and the values shown are theoretical values.

Overall straightness of the derived median line may be verified by designing a gage that encloses the entire shaft length. The gage diameter is equal to the virtual condition of the shaft.

The straightness of the derived median line per unit length may be verified with a functional gage in a similar manner to overall straightness. The gage diameter is calculated to equal the virtual condition across the unit length distance. In the given example, the gage diameter of 1.503" is determined by adding the 1.501" MMC of the shaft and its .002" straightness tolerance.



Goodheart-Willcox Publisher

Figure 5-24. Two functional gages may be used to check the requirements of overall and unit length straightness.

The gage length must equal the specified unit dimension that is shown in the feature control frame. For the given figure, the length is 1.00". The full length of the shaft must slip through the gage without binding or getting stuck. This verifies the .002" tolerance zone at all 1.00" lengths along the shaft.

Straightness of a Derived Median Line between Flat Surfaces

Prior to 2009, a straightness tolerance applied on a thickness dimension was a means to specify the flatness of a derived median plane. The logic was that straightness applied in such a manner did not have direction and would therefore establish a flatness requirement. In the 2009 standard, a change was made and flatness is now applied to define the flatness requirement for a derived median plane. With this change in the standard, it opens the possibility for application of straightness on a derived median line. However, this new application of straightness was not added to the standard.

The following is a possible, yet not standardized, application of straightness tolerance on a derived median line. This practice, if used, should be supplemented with a drawing note that explains the intended requirement. Because this application is not covered by a national standard, the effect on Rule #1 by the application of straightness to a derived median line is uncertain. It is possible

that it *may* completely eliminate the requirements of Rule #1, or it *may* only remove Rule #1 in the direction of the straightness tolerance. To avoid the risks of this ambiguity of straightness of a derived median line, it is important to include additional tolerances or noted requirements to ensure that form tolerances are defined in directions other than the applied straightness tolerance. Derived median plane flatness, surface flatness, and surface profile may be used depending on the needed feature tolerances.

When adequately explained by a drawing note or an internal company standard, a straightness tolerance may be applied to a regular feature of size that contains parallel straight line elements. See [Figure 5-25](#). The feature control frame is placed adjacent to the dimension value or on the dimension line. This indicates, with adequate noted explanation, that each derived median line must be straight within the given tolerance value in the direction of the straight surface elements. The direction of each line element, in the given figure, is parallel to the plane containing the feature control frame.

The actual local size between the line elements may vary within the given size limits and the surface elements may contain form variations provided that the derived median line stays within the specified straightness tolerance. The

straightness tolerance may be applied RFS or at MMC. If no modifier is shown, then the tolerance is understood to be applicable RFS.

In [Figure 5-25](#), a thickness dimension of $.250 \pm .010$ and a straightness tolerance of $.003$ are shown applied to the thickness of the part. The straightness tolerance shows no material condition modifier, so it is applicable RFS. The actual local size may vary between $.240$ minimum and $.260$ maximum. Regardless of the thickness of the feature, the derived median line at each cross section must be within a boundary created by two straight lines separated by $.003$.

The shown straightness tolerance does not specify form of the part in any other direction. Additional requirements must be specified to define requirements in the other directions.

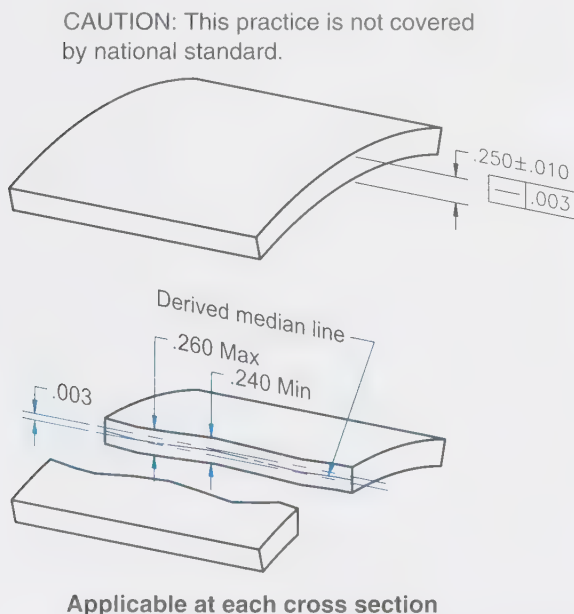
Straightness of a derived median line may be applied with the MMC modifier if the same precautions described above are followed. See [Figure 5-26](#). The straightness tolerance applied to the regular feature of size permits $.003$ variation when the feature is at MMC. The combined effect of the form tolerance and MMC at each cross section results in a virtual condition of $.263$. As the actual local size departs from MMC, the allowed straightness variation of the derived median line increases by an amount equal to the departure from MMC.

Care must be taken to apply straightness tolerance feature control frames in a manner that communicates the allowed extent of variation.

- Placement of the feature control frame on the dimension and applicability at RFS results in allowable variation of the derived median lines within a fixed value straightness tolerance zone. There is no perfect form boundary at MMC.
- Placement of the feature control frame on the dimension and the inclusion of an MMC modifier results in a virtual condition and the tolerance zone size increases as the actual local size departs from MMC. There is no perfect form boundary at MMC.
- Placement of the feature control frame on an extension line from a surface indicates a form tolerance on one surface. The allowable surface variation must be within a perfect form boundary at MMC.

Verification of Median Line Straightness

When the straightness tolerance requirement is specified at RFS as in [Figure 5-25](#), a functional gage of a fixed size is inadequate. When RFS is



Goodheart-Willcox Publisher

Figure 5-25. Derived median line straightness may be tolerated by placing the feature control frame adjacent to the dimension value.

RFS unless an MMC modifier is shown. No datum feature references are shown in a flatness tolerance specification.



Local size	Tolerance
.260	.003
.259	.004
.258	.005
↓	↓
.242	.021
.241	.022
.240	.023

Applied to a Surface

Flatness tolerance specifications applied to a surface include only the flatness tolerance symbol and the tolerance value. See [Figure 5-27](#). Because the controlled feature is a flat surface, no material condition modifier is shown. In orthographic views, the feature control frame is attached to the surface with a leader, or it is placed on an extension line from the surface. In a CAD model, the feature control frame is attached with a leader that terminates with a dot on the surface.

The tolerance zone is bounded by two parallel planes separated by a distance equal to the tolerance value. It is not necessary for the tolerance zone to be in a fixed orientation relative to the part. The tolerance zone may be at any angle provided the part surface does not violate the size limits.

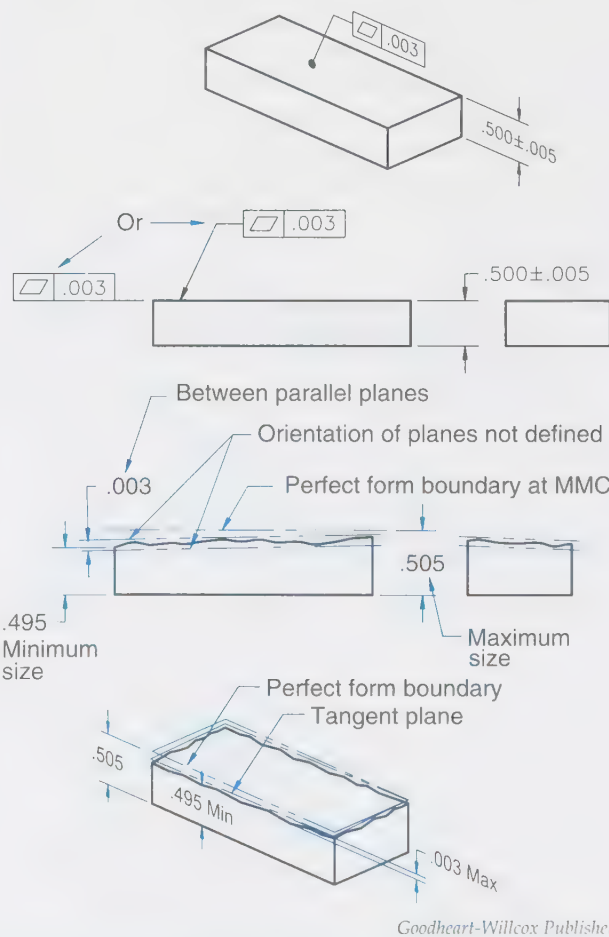


Figure 5-27. Flatness tolerances create a tolerance zone bounded by two parallel planes.

Flatness

Flatness tolerances are used to specify allowable form variation for flat surfaces and for the derived median plane of parallel flat surfaces. No material condition modifier is applicable to flatness when it is specified on a surface. Flatness applied to a regular feature of size is applicable

The requirement for perfect form at MMC (Rule #1) limits the allowable surface flatness variations within the size limits. This means that a flatness tolerance should only be applied if the surface form must be better than is required by the size limits.

The given part has a size tolerance of $\pm .005''$ that results in a total size tolerance of $.010''$. If only the size tolerance were applied to the part, allowable variations on the top surface could be up to $.010''$. However, a flatness tolerance specification of $.003''$ is attached to the top surface to refine the required level of control established by the size tolerance. The surface variations may not exceed $.003''$ relative to a tangent plane that is in contact with the high points on the surface.

As shown in the given figure, one end of the part may be produced at $.495''$, and the other end at $.505''$. These variations in size are within the size tolerance limits. The top surface is also shown to be flat within the $.003''$ tolerance zone. This part configuration is acceptable. Any other configuration that meets both the size limit requirements and the flatness tolerance is also acceptable.

The flatness tolerance specification applies to the entire surface. It establishes a tolerance zone that extends across the surface in all directions. The zone is bounded by two planes. This is different from straightness, which establishes a requirement in only one direction.

The specified $.003''$ flatness tolerance in the given figure is the maximum allowed flatness variation. The entire $.003''$ cannot be used when a part is produced at or near the MMC limit of size, because the part size and form variations must comply with Rule #1.

So, a part that is produced at the $.505''$ MMC cannot use any of the specified flatness tolerance. A part that departed $.001''$ from the MMC size is only permitted $.001''$ form variation. Any form variation greater than $.001''$ would result in violation of the perfect form boundary, and that is not allowed. Likewise, a part produced $.002''$ smaller than MMC is only permitted $.002''$ form variation. Again, when departure from MMC is only $.002''$, the size limits have controlled the allowable amount of form variation. Only when the size departs more than $.003''$ from MMC does the specified form tolerance begin to have an effect.

If a part varied in size $.006''$ from the perfect form boundary, there would be an allowed $.006''$ form variation based only on the size tolerance, but the specified flatness tolerance of $.003''$ must also be met. In this case, the flatness tolerance is what defines the allowable form variation.

So, the size tolerance and Rule #1 must be met, and the flatness tolerance must also be met. The table below shows the combined effects of the size and form tolerances in [Figure 5-27](#).

	Produced Size	Departure from MMC	Specified Flatness	Allowed Form Variation
MMC	.505	.000	.003	.000
	.504	.001	.003	.001
	.503	.002	.003	.002
	.502	.003	.003	.003
	.501	.004	.003	.003
	.500	.005	.003	.003
	.499	.006	.003	.003
	.498	.007	.003	.003
	.497	.008	.003	.003
	.496	.009	.003	.003
LMC	.495	.010	.003	.003

It is important to notice, in the table, that the flatness tolerance specification is unchanged by the change in feature size. The flatness tolerance remains .003 for all sizes of the feature. However, the allowable form variation is controlled by the size limits when the departure from MMC is less than the specified form tolerance value. A part that is produced with a thickness of .504" and has a form variation of .003" is a bad part even though it has met the specified flatness tolerance. It is bad because it has failed to meet the requirement of Rule #1 that requires the feature to be within the perfect form boundary. (The produced .504 size and .003 flatness results in a .507 outer boundary, and that violates the .505 perfect form boundary.) The perfect form boundary requirement only allows a form variation of .001" when the feature is at the .504" size. This effect of Rule #1 should not be confused with the allowable increase in specified tolerance that results from application of the MMC modifier on tolerances applied to features of size.

Verification of Surface Flatness

Flatness of a surface may be verified by running a dial indicator across the surface. See [Figure 5-28](#) and [5-29](#). The part is rested on an

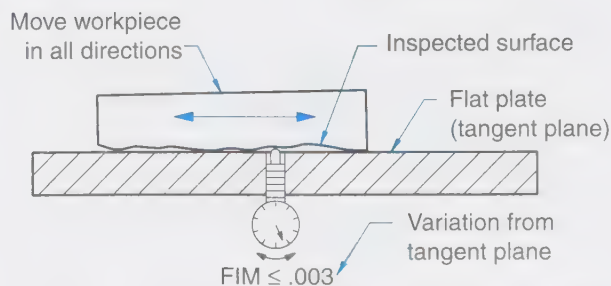
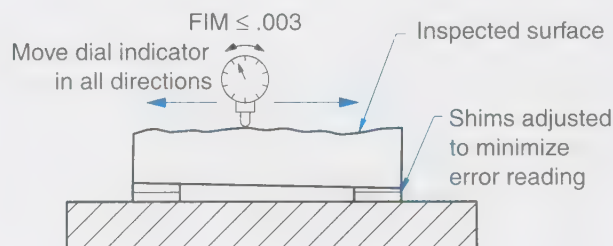
inspection tool such as a surface plate. The dial indicator can be held in a height gage or dial indicator stand and moved across the surface. The height of the dial indicator is not important provided that the indicator stays in one plane as it is moved across the workpiece.

If a flatness tolerance of .003" is specified, then the full indicator movement must be .003" or less. Should the first attempt to verify flatness show too much variation, the part orientation may be changed and another check of the surface made. The orientation should be adjusted to minimize the indicator movement. Part orientation may be adjusted because flatness does not control orientation. Changing the part orientation and repeating the surface measurement is an iterative process.

Adjusting the part to obtain the optimum flatness reading can be difficult. To avoid this problem, another method may be used to check flatness. See [Figure 5-29](#). This inspection process uses a dial indicator installed in a hole on a flat plate. The dial indicator probe is positioned to extend above the surface of the flat plate. The workpiece is placed over the dial indicator probe and in contact with the surface plate. This establishes a plane that is tangent to the surface. The workpiece is moved and the full indicator movement will equal the amount of flatness variation on the surface. The full indicator movement must not exceed the specified flatness tolerance.



Goodheart-Willcox Publisher



Goodheart-Willcox Publisher

Figure 5-28. A dial indicator on a height stand may be used to measure flatness.

Figure 5-29. The orientation of the flatness tolerance zone is not constrained except that size tolerances must be met.

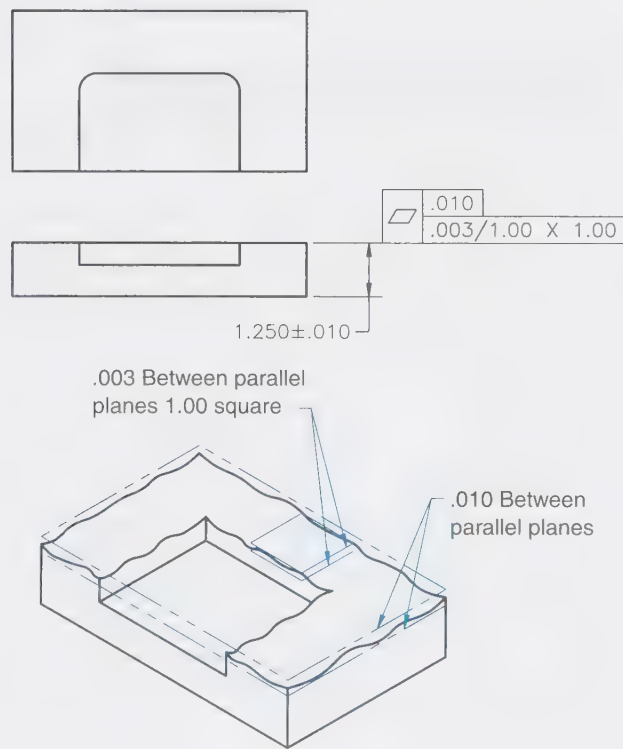
Automated methods may also be used. Data is collected for points on the surface and then an automated process used to best fit the collected points between two parallel planes or fit the parallel planes to the data.

Overall and Unit Area Flatness Tolerance

A flatness tolerance applied to a surface extends across the entire surface. It is possible to apply an overall flatness tolerance in combination with a flatness tolerance per unit area. See **Figure 5-30**. The feature control frame for a combined overall and unit area flatness tolerance specification is completed in a similar manner to the one used for straightness per unit length.

Only one flatness symbol is used in a two-segment feature control frame. The first segment (the upper line) shows the overall flatness tolerance. The lower segment (the second line) shows the flatness per unit area and the size of the unit area. It is common practice to make the unit area square or circular. The feature control frame is applied in the same manner as the previously explained single-segment (one line) flatness tolerance.

In **Figure 5-30**, a flatness requirement of .010" is specified across the top surface of the given part. The requirement only affects the top surface. The



Goodheart-Willcox Publisher

Figure 5-30. Flatness per unit area may be combined with overall surface flatness.

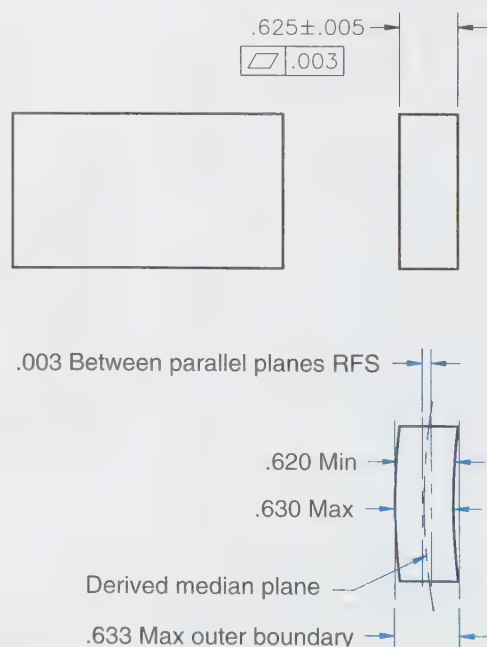
second line of the combined flatness specification requires that flatness be within .003" across any 1.00" square area on the surface. The feature must meet the size requirements in addition to the two flatness requirements.

Flatness Tolerance Applied to a Derived Median Plane

A flatness tolerance may be applied to a regular feature of size defined by parallel flat surfaces. This controls the derived median plane of the feature. See **Figure 5-31**. In orthographic views, the feature control frame is placed adjacent to the dimension value or on the dimension line. Showing the feature control frame in one of these positions indicates that the derived median plane must be flat within the given tolerance value. When a form tolerance is applied to a regular feature of size, the requirements of Rule #1 are not applicable to that feature.

In orthographic views, to establish a tolerance on the derived median plane, care must be used to avoid making contact between the feature control frame and any extension lines from the surfaces. If an extension line is contacted, it will appear the intent is to establish surface flatness instead of derived median plane flatness.

Straightness of the derived median plane is not restricted by the limits of size (Rule #1) when a flatness tolerance is specified on a feature of size. In fact,



Goodheart-Willcox Publisher

Figure 5-31. Derived median plane flatness is tolerated by placing the feature control frame adjacent to the dimension value.

Rule #1 does not apply when flatness tolerance is applied to a feature of size.

History Brief

The ASME Y14.5M-1994 and earlier standards established control of a derived median plane using a straightness tolerance.

The method used in a CAD model is to show the feature control frame adjacent to the size dimension. The placement near the size dimension and associativity with the feature is the indicator that the tolerance applies to the derived median plane and not to the surfaces.

Application of a flatness tolerance to a regular feature of size does not directly control the surface conditions. The surfaces may vary to any size within the given size limits if the derived median plane stays within the specified flatness tolerance.

When a flatness tolerance is applied to a flat regular feature of size, the tolerance is assumed to apply at RFS. If it is desired to apply the tolerance at MMC, the modifier must be applied.

Flatness of a derived median plane, when applied RFS, creates a boundary equal to the MMC size and the effect of the tolerance. The given figure has a maximum size limit of .630" and the flatness tolerance is applicable RFS. The flatness tolerance applied to the regular feature of size permits .003" variation when the feature is at any allowable size. The combined effect of the form tolerance and the maximum size is a maximum outer boundary of .633".

Pro Tip

Tolerancing a Derived Median Plane

Care must be taken to apply flatness tolerances in a manner that achieves the desired control. Placement of the feature control frame on the dimension results in a tolerance zone for the derived median plane.

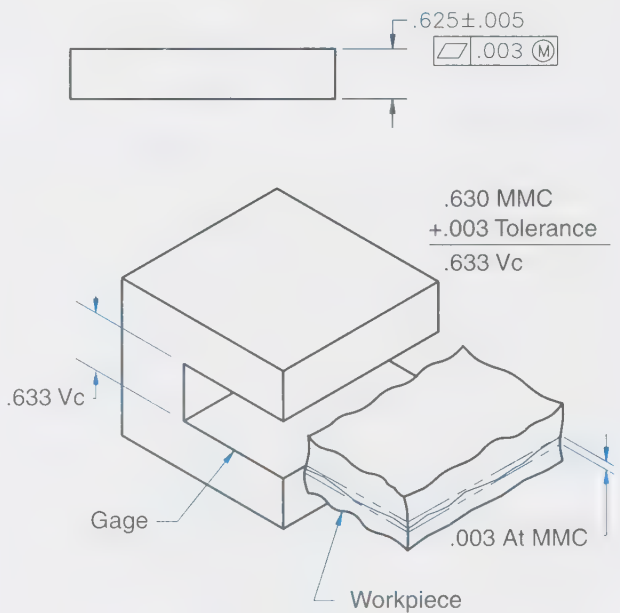
Care must be taken to apply flatness tolerance feature control frames in a manner that communicates the desired tolerance zone.

- Placement of the feature control frame on the dimension and applicability at RFS results in an allowable variation of the derived median plane within a fixed value flatness tolerance zone. There is no perfect form boundary at MMC.

- Placement of the feature control frame on the dimension and the inclusion of an MMC modifier results in a virtual condition and the tolerance zone size increases as the actual local size departs from MMC. There is no perfect form boundary at MMC.
- Placement of the feature control frame on an extension line from a surface indicates a form tolerance on one surface. The allowable surface variation must also be be within a perfect form boundary at MMC.

It is permissible to apply a flatness tolerance that is larger than the size tolerance on a feature of size. This does not require an exception to Rule #1 be noted because a flatness tolerance applied to a feature of size overrides the requirements of Rule #1.

A flatness tolerance applied to a feature of size is assumed to apply RFS unless otherwise shown. A flatness tolerance with an MMC modifier creates a derived median plane tolerance zone and virtual condition boundary that extend across the entire feature of size. See Figure 5-32. The tolerance zone



Local Size	Tolerance
.630	.003
.629	.004
.628	.005
.622	.011
.621	.012
.620	.013

Goodheart-Willcox Publisher

Figure 5-32. A flatness tolerance applied to control the derived median plane at MMC can be verified with a functional gage.

in the given figure is .003" wide when the feature is at MMC. The zone width is permitted to increase as the feature size departs from MMC.

Verification of Derived Median Plane Flatness

Flatness of a derived median plane may be inspected with a functional gage when the flatness tolerance specification includes the MMC modifier. The theoretical gage must simulate the virtual condition of the feature that is inspected. See Figure 5-32.

The given figure has a flatness tolerance of .003" at MMC. This value plus the MMC size of .630" results in a virtual condition of .633". The gage to check the derived median plane flatness for the given part must have a slot size of .633". The gage slot must be large enough to totally enclose the entire feature being inspected.

If the flatness tolerance requirement is specified at RFS, then a functional gage of a fixed size is not adequate. When RFS is applicable, individual readings can be taken on both sides of the part and the derived median plane location calculated as was done for derived median line straightness in Figure 5-22.

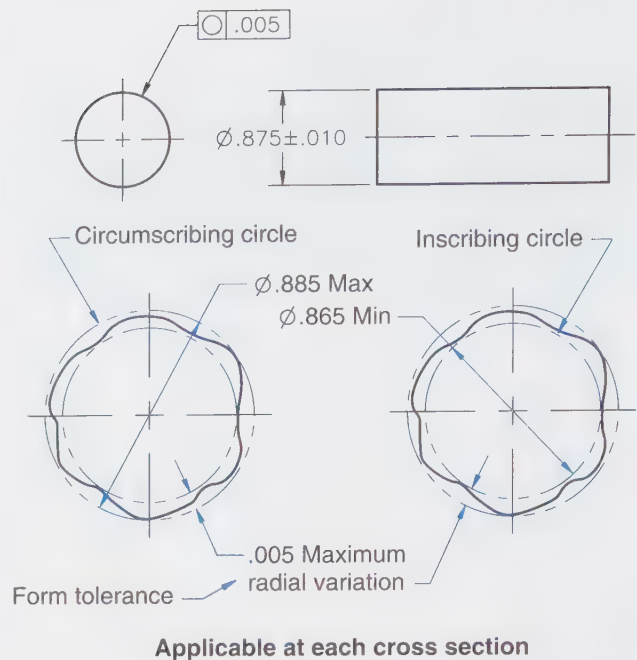
Circularity

Circularity tolerances establish the allowable variation on the roundness of any feature that has a circular cross section, including cylinders, cones, and spheres. Although a cone does not have a constant diameter, it does have a circular cross section. Circularity tolerances are often referred to as roundness tolerances, but roundness is not the preferred or correct term.

Applied to a Cylindrical Surface

Circularity is only applied as a control of surface form and establishes a requirement that applies at each cross section of the feature. See Figure 5-33. The feature control frame is always attached to the surface of the part using a leader and not associated with the size dimension. Because the tolerance is not attached to a size feature, no material condition modifier is applicable. The tolerance is most clearly specified when a leader is used to attach the feature control frame to a circular view. The tolerance value is the width of the tolerance zone, meaning the radial distance between the circles bounding the tolerance zone. The tolerance value is not the diameter difference between the circles.

A diameter dimension on a cylinder establishes a perfect form boundary at MMC. Circularity



Goodheart-Willcox Publisher

Figure 5-33. Circularity tolerances establish the allowable variation on circular cross sections.

variation is only permitted when the size has departed from the MMC perfect form boundary. This means that circularity variations cannot be greater than is permitted by the size tolerance.

The given figure has a perfect form boundary at .885", the MMC size. Based on only the .865" minimum size limit, it would be allowable for surface variations as large as .020" to occur inside the perfect form boundary. The .020 form variation on one side of the shaft is possible because no requirement exists for the smallest size limit to remain centered on the axis of the perfect form boundary. If there is a need for circularity to be better than the .020" variation, then a circularity tolerance may be applied.

The tolerance zone boundaries for a circularity tolerance are two concentric circles. The outer circle may be the minimum circumscribing circle, or the inner boundary may be the maximum inscribing circle. Once the outer or inner boundary is established, the second boundary must be concentric with the first boundary. The maximum allowable diameter of the circumscribing circle on an external feature of size such as a shaft is the MMC size. The circularity tolerance value is the radial distance between the outer boundary circle and its concentric inner boundary circle. The circularity tolerance value is *not* the diameter difference. The radial difference between the circles is half the diameter difference. This means that a circularity tolerance specification of .005" is bounded by two circles that have a diameter difference of .010".

The circularity tolerance boundary circles may be any size provided they are separated by the tolerance value and the feature is also within size limits. The tolerance boundaries may be best fit to the surface, or they may be established in a manner that fits one of the boundaries to the surface.

When fitting either the inner or outer boundary to the surface, it is important to know that only one of them will provide the best results. The best results can only be determined by trying both approaches. If the surface checks good when best fitting the boundaries or when using either of the following approaches, the surface is acceptable.

Approach 1: At each cross section, the outer boundary may be established using the smallest circumscribing circle. The inner boundary is then established concentric to the outer boundary at a radial distance equal to the circularity tolerance value. If all points on the surface are within the two boundaries, the feature is good at this location. If all points are not within the boundaries, attempting a different approach is necessary to determine if the surface is acceptable.

Approach 2: At each cross section, the inner boundary is established using the maximum inscribing circle. The outer boundary is then established concentric to the inner boundary and at a radial distance equal to the circularity tolerance value. If all points on the surface are within the two boundaries, the feature is good at this location.

To establish the actual circularity variation at each cross section, the above two approaches may be used with measurements being made relative to the smallest circumscribing and largest inscribing circle. Whichever provides the smallest variation measurements is the one that determines the actual amount of circularity variation at that cross section.

The given part has a perfect form boundary of .885" diameter. The largest allowed circumscribing circle is equal to the MMC size. In addition to all surface variations remaining within this boundary, all diameter measurements across the part must be within the size limits of .865" and .885".

In addition to meeting the size requirements as described in the previous paragraph, the surface variations must be contained within the circularity tolerance boundary created by two concentric circles separated .005" radially. The circles forming the boundary are concentric, but are not required to be centered on the axis of the shaft. The tolerance zone boundaries may be any diameter, provided they are separated by .005" and the part surface does not violate the size dimension limits.

The circularity tolerance defines the allowable form variations of the surface at each cross section. The variations at each cross section are considered separately from all others. It is possible that two cross sections meet the circularity tolerance, and also have the tolerance zone at those cross sections offset in different directions from the axis.

Application to an internal feature such as a hole is very similar to the application on an external feature. The difference is that the MMC size is the smallest allowable size.

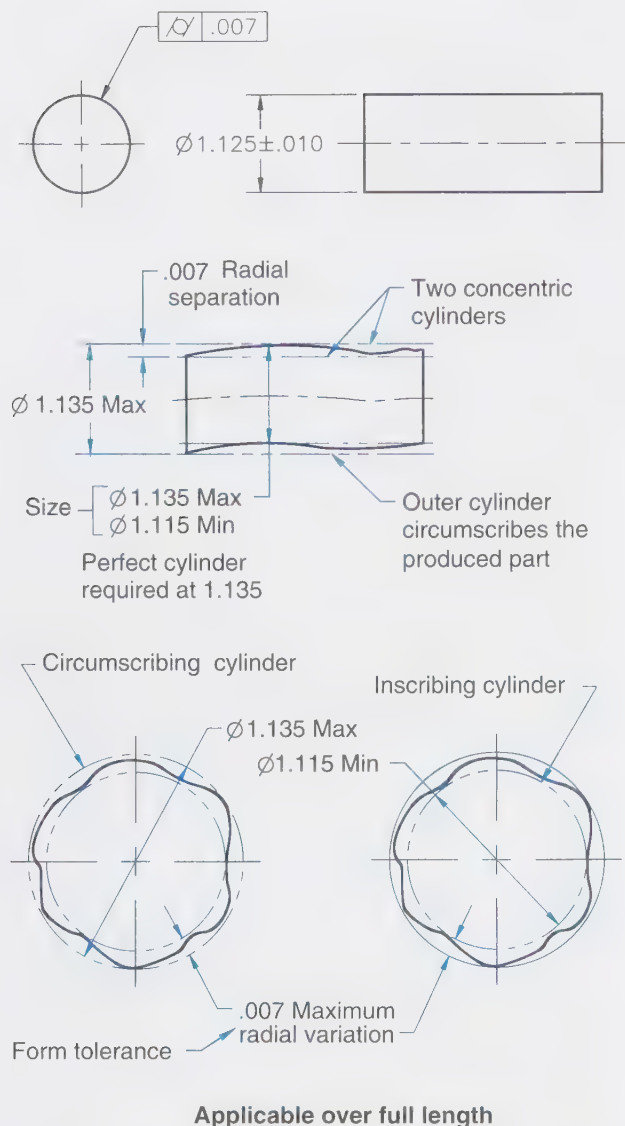
Applied to a Cone

The form of the circular cross section of a cone may be controlled with a circularity tolerance. The tolerance zone at each cross section is bounded by two concentric circles. However, the tolerance zone boundary diameters change according to the location along the cone axis. Circularity has no effect on the control of the cone angle or the cone size limits. It only controls the circular cross section. The feature control frame may be attached to the surface using either a leader or placement on an extension line. Typically, profile is the preferred means of specifying form tolerance on a cone.

Cylindricity

Allowable form variation of a cylindrical surface may be specified by a *cylindricity* tolerance that extends the entire length of the feature. It may be applied to a shaft or hole. See [Figure 5-34](#). This tolerance type is, in effect, a combination of circularity and straightness tolerances. It also controls parallelism of the sides of the cylinder to prevent the part from being tapered. Material condition modifiers are not included in the feature control frame because cylindricity is a surface form tolerance.

The tolerance zone for a cylindricity tolerance is bounded by two concentric cylinders. The outer cylinder may be the minimum circumscribing cylinder, or the inner boundary may be the maximum inscribing cylinder. Once the outer or inner boundary is established, the second boundary must be concentric with the first boundary. The maximum allowable diameter of the circumscribing cylinder on an external feature of size such as a shaft is the MMC size. The cylindricity tolerance value is the radial distance between the outer boundary cylinder and its concentric inner boundary cylinder. The cylindricity tolerance value is *not* the diameter difference. The radial difference between the cylinders is half the diameter difference. This means that a cylindricity tolerance specification of .007"



Applicable over full length

Goodheart-Willcox Publisher

Figure 5-34. Cylindricity tolerances result in a tolerance zone bounded by two concentric cylinders.

is bounded by two cylinders that have a diameter difference of .014".

The cylinders bounding the tolerance zone may be any diameter provided they are separated by the tolerance value and the feature is within size limits. The tolerance boundaries may be best fit to the surface, or they may be established in a manner that fits one of the boundaries to the surface.

When fitting either the inner or outer boundary to the surface, it is important to know that only one of them will provide the best results. The best results can only be determined by trying both approaches. If the surface checks good when best fitting the boundaries or when using either of the following approaches, the surface is acceptable.

Approach 1: The outer boundary may be established using the smallest circumscribing cylinder.

The inner boundary is then established concentric to the outer boundary at a radial distance equal to the cylindricity tolerance value. If all points on the surface are within the two boundaries, the feature is good. If all points are not within the boundaries, another check of the cylindricity is required to determine if the surface is acceptable.

Approach 2: The inner boundary is established using the maximum inscribing cylinder. The outer boundary is then established concentric to the inner boundary and at a radial distance equal to the cylindricity tolerance value. If all points on the surface are within the two boundaries, the feature is good.

To establish the actual cylindricity variation, the above two approaches may be used with measurements being made relative to the smallest circumscribing and largest inscribing cylinder. Whichever provides the smallest variation measurements is the one that determines the amount of cylindricity error.

The given figure shows a shaft with a 1.135" maximum size, which is the diameter of the perfect form boundary. The minimum size limit of 1.115" establishes the smallest permitted cross-sectional measurement. Because there is a cylindricity tolerance of .007", there cannot be cross-sectional measurements at both limits of size. So, the cylindricity tolerance does not establish the size limits, but it does establish an allowable amount of size and form variation within the size limits.

If the largest cross-sectional measurement is 1.132", then the diameter of the cylinder forming the outside of the cylindricity tolerance must be 1.132". The diameter of the inside cylinder of the tolerance zone is .014" less than the large zone (.007" radially). This means the inside cylindricity tolerance boundary has a 1.118" diameter. No part of the surface may extend inside this cylinder.

The requirements of the size limits and the form tolerance must both be met. It is not possible to have cross-sectional measurements of 1.135" and 1.115" for the given part, because these measurements would mean the form tolerance (cylindricity) has been violated. The cylindricity tolerance prevents the feature size varying across the entire size range on any individual part. However, it is acceptable for one part to measure at the 1.135" maximum size and another part to measure at the 1.115" minimum size. Any measurements greater than 1.135" or less than 1.115" are unacceptable regardless of how perfect the cylindrical shape.

Cylindricity tolerances may only be applied to cylindrical features such as holes and shafts. They are not applicable to any other features.

Chapter Summary

- ✓ The four form tolerances are straightness, flatness, circularity, and cylindricity.
- ✓ Straightness and flatness are the two form tolerances that may be applied to a regular feature of size.
- ✓ Limits of size define allowable form variations. There is a perfect form boundary at the maximum material condition.
- ✓ The perfect form boundary at MMC does not control the orientation or relationships between multiple features.
- ✓ Form tolerances are assumed to apply at RFS when applied to features of size.
- ✓ The way in which a straightness or flatness feature control frame is applied to a part indicates whether the surface or regular feature of size is tolerated.
- ✓ Straightness, when applied to a surface, controls single line elements in one direction.
- ✓ Flatness applied to a surface creates a tolerance zone bounded by two parallel planes.
- ✓ Circularity may be specified on any feature that has a circular cross section.
- ✓ The circularity tolerance zone is bounded by two concentric circles.
- ✓ The circularity tolerance value is the radial distance between the concentric circles that form the tolerance zone.
- ✓ Cylindricity may only be specified on cylindrical features.
- ✓ The cylindricity tolerance zone is bounded by two concentric cylinders.
- ✓ The cylindricity tolerance value is the radial distance between concentric cylinders that form the tolerance zone.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. ____ is not a form tolerance.
A. Perpendicularity
B. Straightness
C. Flatness
D. Cylindricity
2. A feature control frame for a form tolerance must contain a ____.
A. datum feature reference
B. diameter symbol
C. tolerance value
D. All of the above.
3. Each form tolerance specification applied on an extension line ____.
A. only controls surface conditions
B. controls a derived median line or derived median plane
C. may control two surfaces
D. is a drawing error
4. A feature must have perfect form when ____.
A. it is at MMC
B. it is at LMC
C. form tolerances are not specified
D. the form tolerance is specified without an MMC modifier
5. A ____ tolerance specification defines the allowable form variation of line elements in only one direction.
A. circularity
B. cylindricity
C. flatness
D. straightness
6. A flatness tolerance results in a boundary defined by two ____.
A. concentric circles
B. irregular curves
C. parallel lines
D. parallel planes
7. Derived median line straightness for a cylinder may be specified on a drawing by placing the feature control frame ____.
A. adjacent to an extension line
B. on the centerline
C. adjacent to the diameter dimension
D. None of the above.
8. ____ may be used to control the cross-sectional shape of a cone.
A. Circularity
B. Straightness
C. Cylindricity
D. Flatness
9. A tolerance zone bounded by two concentric cylinders is established when ____ specified.
A. cylindricity is
B. circularity and straightness are
C. circularity is
D. Both A and B.

10. A form tolerance only need be applied to a surface if the needed amount of surface form tolerance is _____ the amount of the size tolerance.
 - A. more than
 - B. equal to
 - C. less than
 - D. unrelated to
11. Verification of a form tolerance is _____ if the tolerance is applied to a regular feature of size and has an MMC modifier.
 - A. ignored
 - B. checked only with coordinate measuring machines
 - C. not necessary at MMC
 - D. possible to be checked with functional gages
12. Application of a straightness tolerance on a cylindrical feature of size results in a(n) _____.
 - A. impossible to check part
 - B. virtual condition
 - C. tolerancing error
 - D. envelope equal to the MMC size
13. A cylindricity tolerance specification defines the _____ distance between two concentric cylinders that define the tolerance boundary.
 - A. radial
 - B. diameter
 - C. cone
 - D. axial
18. *True or False?* An MMC modifier is permitted on a form tolerance only if the tolerance is applied to define a requirement for the derived median line or derived median plane of a regular feature of size.
19. *True or False?* A form tolerance must be smaller than the size tolerance only if the feature is a cylinder.
20. *True or False?* Two flatness tolerance requirements, in a two-segment feature control frame, may be placed on one surface if one tolerance is applied to the entire surface and a smaller tolerance is applied to unit areas on the surface.
21. *True or False?* The concentric circles that form a circularity tolerance zone have a diameter difference equal to the tolerance value.

Fill in the Blank

True/False

14. *True or False?* Cylindricity tolerances are always required because the size tolerance on a diameter dimension does not establish any requirement for the shape of the cylinder.
15. *True or False?* Bar stock is not required to have perfect form at MMC, but thin-walled tubing and sheet stock must have perfect form at MMC.
16. *True or False?* Surface variations on the top surface of a flat plate must not be greater than the size tolerance.
17. *True or False?* A derived median line straightness tolerance on a hole results in a virtual condition that is smaller in diameter than the MMC of the hole.
22. Size dimensions establish requirements for the size and _____ of the dimensioned feature.
23. _____ and flatness form tolerances may be applied to features of size.
24. The MMC size of a hole is the _____ limit of size for the hole.
25. _____ may be used to specify the allowable straightness variation for a specific unit length on a shaft.
26. Verifying that parallel line elements are straight in one direction on a surface does not verify flatness, because flatness requires measurements in more than one _____.
27. The _____ symbol must be placed in front of the tolerance value when straightness is applied to establish a derived median line straightness tolerance on a shaft or hole.
28. The abbreviation FIM stands for _____.
29. A form tolerance applied to a flat surface does not require a specific _____ to any other feature on the part.
30. A feature control frame is placed _____ to indicate a surface requirement.

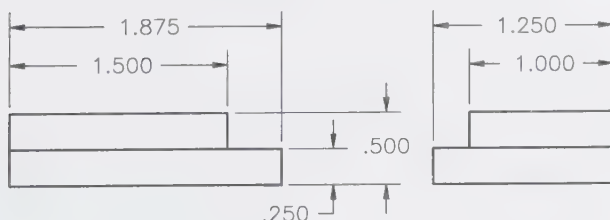
Short Answer

31. Describe how the virtual condition for a shaft is calculated when the shaft has a derived median line straightness tolerance applied and the tolerance specification includes an MMC modifier.
32. Name the four form tolerances.
33. Explain why form tolerances are used instead of reducing size tolerances to control surface variations.
34. Explain what must be applied to a size dimension when a form tolerance greater than the size tolerance is being applied to a surface.
35. An MMC modifier may only be used on a form tolerance if it is applied to which kind of feature?
36. Explain what a bonus tolerance is.
37. Sketch the feature control frame to show a straightness tolerance that requires an .008" diameter zone when the part is produced at MMC.
38. Sketch a feature control frame that requires a surface flatness of .011".

Application Problems

Some of the following problems require that a sketch be made. All sketches should be neat and sufficiently accurate to clearly show your answer. Each problem description requires the addition of some dimensions for completion of the problem. Apply all required dimensions in compliance with dimensioning and tolerancing requirements.

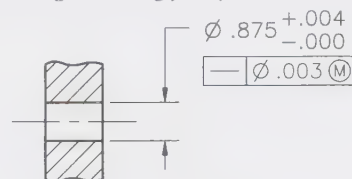
39. Apply a feature control frame to specify a flatness tolerance for the top surface of the given part to .006".



40. Apply a derived median line straightness tolerance of .012" diameter on the .563" diameter. Also apply a derived median line straightness tolerance of .008" on the .785" diameter. Show the MMC modifier on both tolerances.

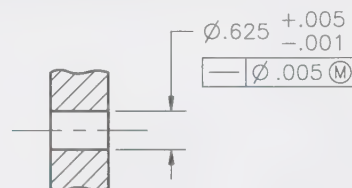


Use the following drawing for questions 41, 42, and 43.



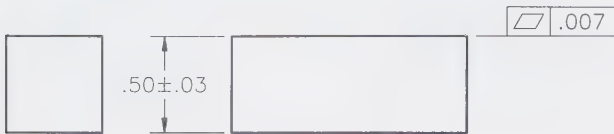
41. Using the given hole specification, calculate the virtual condition of the hole. Show your calculations.
42. The given hole specification permits .003" derived median line straightness variation at MMC. Calculate the allowable straightness tolerance for a hole produced at .877" diameter. Show your calculations.
43. What is the MMC size for the given hole?

Use the following drawing for questions 44, 45, and 46.



44. Using the given hole specification, calculate the virtual condition of the hole. Show your calculations.
45. The given hole specification permits .005" derived median line straightness variation at MMC. Calculate the allowable straightness tolerance for a hole produced at .629" diameter. Show your calculations.
46. What is the MMC size for the given hole?

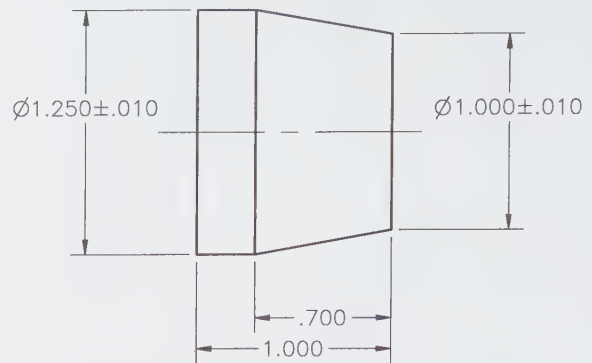
47. Sketch the given part to show the possible surface conditions for the top surface. Show the size tolerance and form tolerance zones.



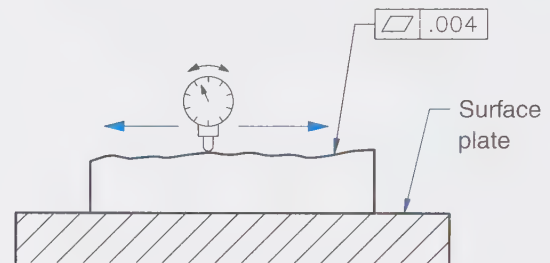
48. Sketch one possible effect of the given tolerances. Show the virtual condition boundary and show the derived median line straightness tolerance zone.



49. Show a straightness tolerance of .009" for the cone elements.



50. The first attempt to verify the required .004" flatness tolerance on the given part resulted in readings of .005". The orientation of the part was adjusted, and the second attempt to check the part resulted in a reading of .003". Explain whether or not the part is good.



Chapter 6

Datums and Datum Feature References

Objectives

Information in this chapter will enable you to:

- ▼ Define the difference between a theoretically perfect datum and a datum feature.
- ▼ Explain how to create a datum reference frame through references made in a feature control frame.
- ▼ Utilize all methods for identifying datum features, including the use of target points, lines, and areas.
- ▼ Make datum feature references in a feature control frame using the correct order of precedence.
- ▼ Explain how a datum reference frame may be simulated when three mutually perpendicular surfaces are referenced as datum features.
- ▼ Use material boundary modifiers on datum feature references and explain the significance of the modifiers.
- ▼ Identify the degrees of freedom constrained by each referenced datum feature in a datum reference frame.

Technical Terms

clocking
compound datum feature reference
compound datum features
datum
datum features
datum feature references
datum feature symbol
datum precedence
datum reference frame (DRF)
datum simulation
datum simulators
datum target symbol

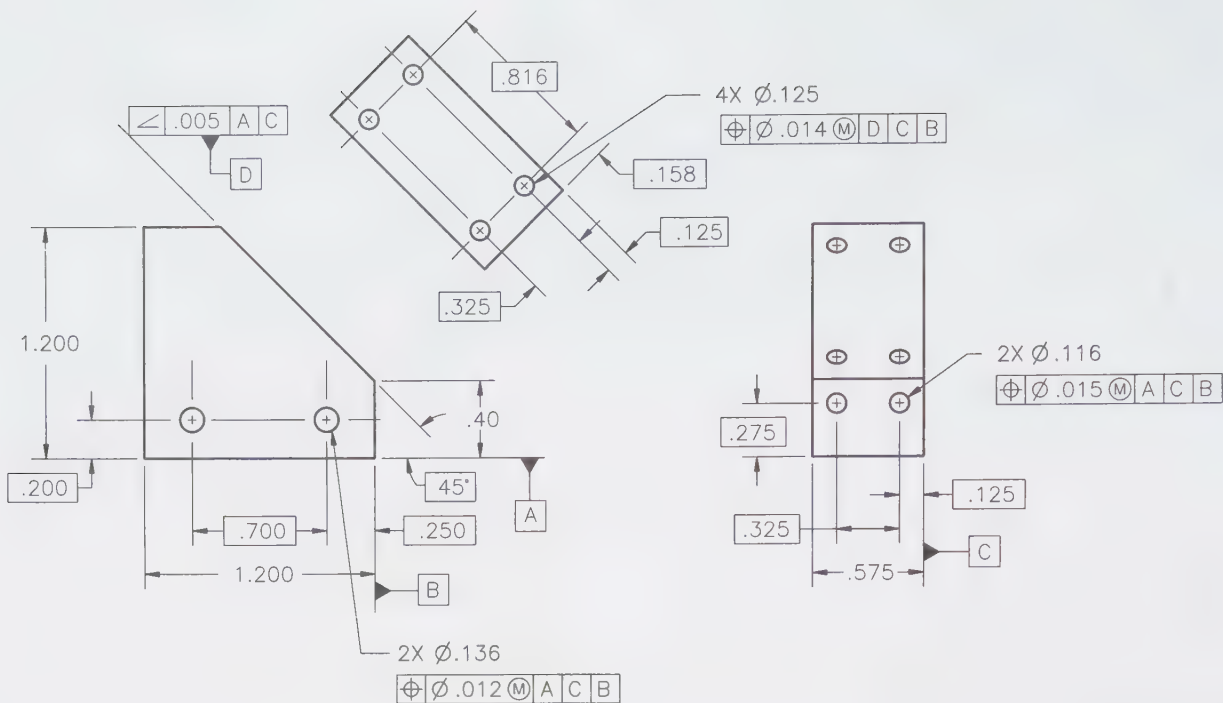
degrees of freedom
envelope
equalizing datum targets
least material boundary (LMB)
least material condition (LMC)
least material size (LMS)
maximum material boundary (MMB)
movable datum target symbol
related actual mating envelope
stepped datum surfaces
target areas
target lines
target points
true geometric counterpart (TGC)
unrelated actual mating envelope

Introduction

Tolerance specifications may be used to define relationships between features. Examples of tolerance specifications that define relationships between features include position, orientation, and profile tolerances.

The feature control frames that define these types of tolerances include datum feature references. See **Figure 6-1**. The orientation and position tolerances in the figure each include datum feature references. Those references indicate how to establish a coordinate system for measuring the location of the tolerated features. The datum feature references (in the feature control frames) indicate which datum features (features on the part) establish the theoretical datums (planes that establish the coordinate system) from which measurements are made.

Three similar sounding but significantly different terms appear in the preceding paragraph. These terms are *datum feature references*, *datum features*,



Goodheart-Willcox Publisher

Figure 6-1. Datums and datum feature references make it possible to define relationships between features.

and *datums*. It is important to understand the difference between them.

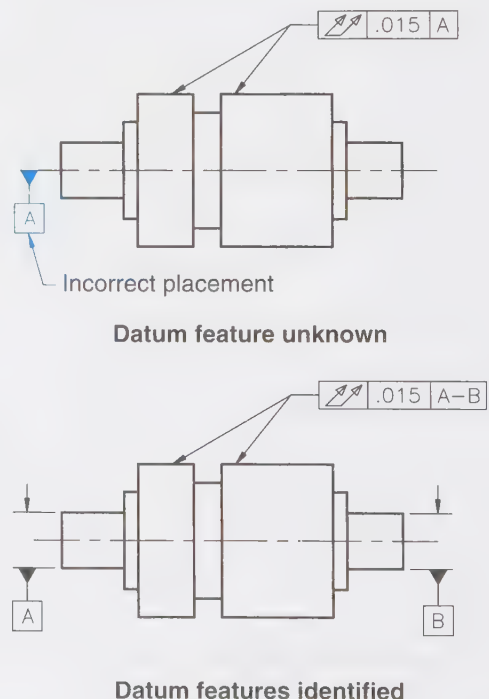
Datum feature references are the letters shown in a feature control frame or possibly in a drawing note. They tell us what features on the part to use in establishing the datum reference frame (coordinate system).

Datum features are physical features on a part. Datum features are identified on a drawing by the use of datum feature symbols or datum targets. A hole is an example of something that can be used as a datum feature because it is something that can be seen or touched. Centerlines, center planes, and theoretical axes are not identified as datum features because they do not physically exist.

A datum is different than a datum feature. A **datum** is a theoretically perfect point, line, or plane. These theoretical geometric entities are located by the physical datum features that are identified on the drawing. As an example, a datum axis may be established from a hole that is a datum feature. The tolerance requirement in the feature control frame is actually located or oriented to the theoretical datums.

The question often arises as to why datum features are needed. Why not simply identify the datum, such as a datum axis? The reason is that an axis, or datum plane, may pass through multiple features. It would not be clear how a datum axis is to be established when it passes through a

shaft that has multiple diameters. To avoid confusion, the specific diameter (or diameters) that establish the datum axis are identified as a datum feature. See **Figure 6-2**. The upper segment of the figure shows a datum feature symbol incorrectly



Goodheart-Willcox Publisher

Figure 6-2. Datum feature identification provides a clear definition of what features to use when establishing a datum.

applied on an axis. There are numerous features and combinations of features that could be used to establish an axis. Only one method is correct for the part's function. The lower segment of the figure shows datum features applied to two diameters. From the feature control frame, it is known that both diameters are to be used in establishing the axis.

Although datums cannot be touched or seen, they are an important part of tolerance application and verification. Tolerance specifications reference datum features and the tolerances are related to theoretical datums established from those datum features. Datum features are identified on the physical elements of the part to define the features that are used in the location of the datums from which to make measurements. In this text, there are explanations of tolerance zones that are related to referenced datums or datum reference frames. The *referenced datums* are the *theoretical datums* established from the *datum features* in the order of precedence defined by the *datum feature references* that are shown in a feature control frame or note.

Because measurements are made with respect to the theoretical datums, the datums must be established relative to the physical features of the part in some manner. The theory will be explained first, then a practical application will be explained. The practical approach is intended to provide a means of putting the theory into practice.

In theory, a datum is established by relating a true geometric counterpart to a datum feature. A **true geometric counterpart (TGC)** is a perfect geometric shape, or a theoretically perfect simulator, corresponding to the shape of a datum feature on a part. As an example, a planar simulator may be brought into contact with a flat surface on a part to establish the location of the datum plane. Another example is a theoretical cylinder that is brought in contact with a shaft to establish a datum axis. True geometric counterparts exist only in theory. The ASME Y14.5-2009 standard replaced the term true geometric counterpart with the word simulator, meaning a theoretically exact datum feature simulator. That seems to have created confusion between a simulator that is theoretically exact and a simulator that is real and imperfect. To avoid the confusion between perfect and imperfect simulators, the term true geometric counterpart will be used here. A true geometric counterpart used to establish a datum is the same as a theoretically perfect datum simulator within this textbook.

In practical application, datum features are simulated using tooling features or they are generated from measurement data using mathematical

processes. Datum simulators used in the figures of this book are shown as tooling features and no tolerances are applied to them. The process of establishing the datums from the imperfect datum features is referred to in this text as **datum simulation** (or simulation) and the tools and processing equipment are referred to as *datum simulators*. The **datum simulators** (whether a physical tool or computer generated) provide a surface or machine-generated plane, axis, or point from which measurements may be made.

Datum Reference Frame

Orientation, runout, and concentricity tolerances always include references to datum features. Profile tolerances may or may not include datum feature references, depending on the required level of control. Position tolerances almost always require datum feature references, but may be specified without them in a very small number of special applications. The exceptions are explained in the chapters covering position and profile.

Datum feature references are shown on the right end of the feature control frame. See **Figure 6-3**. Depending on the tolerance specification and the needed level of control, there may be one, two, or three datum feature references.

The shown order of letters in the feature control frame signifies **datum precedence**. The letter of the alphabet used to identify a datum does not affect precedence.

Datum feature references shown in a feature control frame establish a datum reference frame. See **Figure 6-4**. A **datum reference frame (DRF)** is made up of three mutually perpendicular planes. These planes locate the three axes that form an X, Y, and Z coordinate system.

The X, Y, and Z axes may be identified on the drawing, and identifying them is required if the datum feature references include degrees of freedom

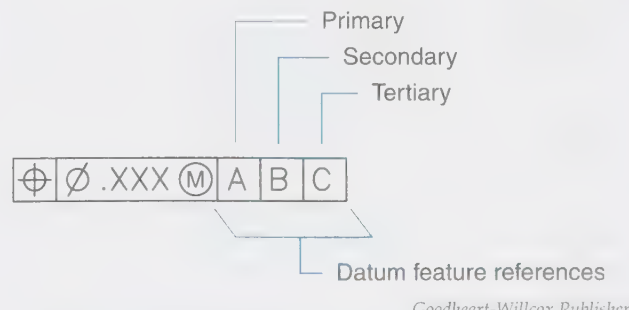


Figure 6-3. Datum feature references are included on the right end of the feature control frame. They are read from left to right.

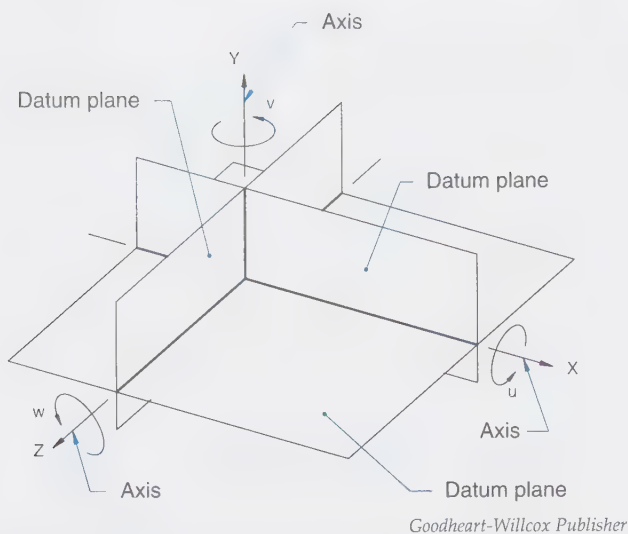


Figure 6-4. A datum reference frame is theoretically perfect and is made of three mutually perpendicular planes.

notations called a customized datum reference frame. Customized datum reference frames are explained later in this chapter. Uppercase letters are used to identify the axes. See [Figure 6-5](#).

Planes in a datum reference frame are always mutually perpendicular and perfect. Although a perfect datum reference frame is located by imperfect datum features, the imperfections in the features are not permitted to affect the datum reference frame. The planes of the datum reference frame remain mutually perpendicular regardless of the conditions of the datum features. This is important, because it prevents the variation in the datum features from affecting the measurement of the toleranced features.

Datum feature references shown in a feature control frame communicate which part features are to be used to establish the datum reference frame. See [Figure 6-5](#). Selection of the features to use is primarily driven by the functional requirements of the part, but the impact of datum feature selection on fabrication and inspection methods must also be considered.

Three surfaces on the given part are selected to act as datum features A, B, and C. A feature control frame shows the position tolerance that is applied to the two holes. Datum features A, B, and C are referenced in the feature control frame. The tolerance specification, as shown, requires that the three datum features be used to locate a theoretically perfect datum reference frame. How they are used will be explained in following paragraphs.

There are advantages to having a perfect datum reference frame from which to make measurements. One advantage is that surface variations

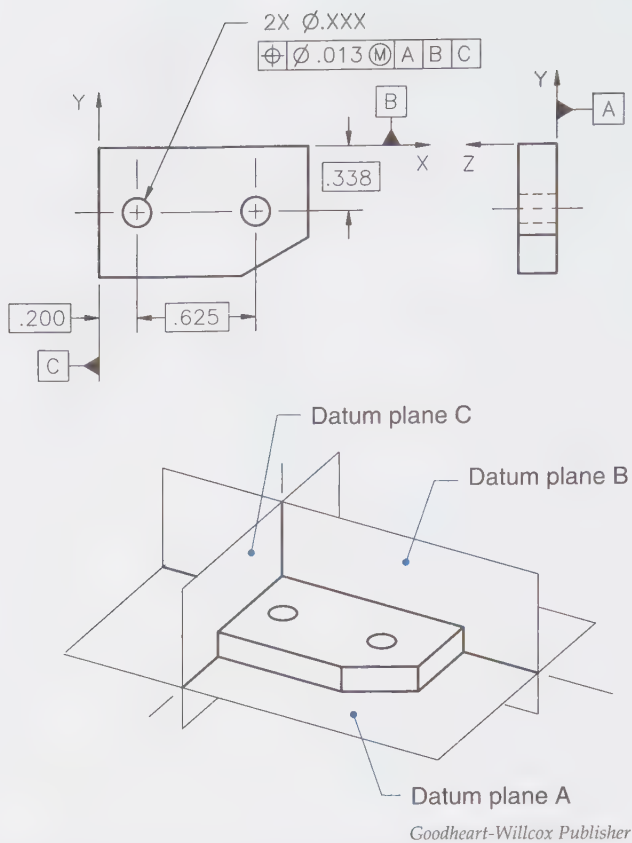
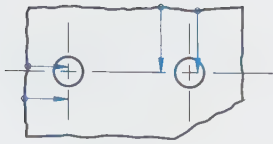


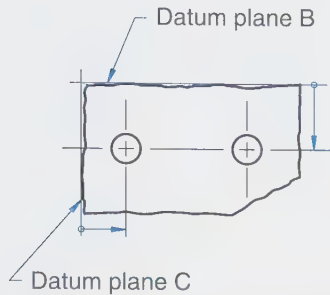
Figure 6-5. Datum feature references made in a feature control frame determine how a part is located in the datum reference frame.

on the part will not impact the locations of holes. Another advantage is that measurements may always be made relative to a known coordinate system. This is in contrast to making measurements from surfaces that may not be perfectly perpendicular.

A drawing of a fabricated part similar to the one in [Figure 6-5](#) is shown in [Figure 6-6](#). The upper segment of the figure shows that questions arise regarding the origin for measurements when a drawing shows no datum features and has no tolerances referencing the datum features. It is unknown whether the measurements should come from the high or low points on the edges of the part. There are no good answers because there is no standard to define the meaning when datums and tolerances are not correctly specified. The lower segment of the figure shows the same part and the correct measurement origin when datum features and tolerances are specified as shown in [Figure 6-5](#). The measurements all originate from the datum reference frame. Identifying datum features and specifying tolerances that reference the datum features is the only way to specify requirements that have only one correct interpretation.



Measurement origin undefined
without datums



Measurement origin well defined
with datums

Goodheart-Willcox Publisher

Figure 6-6. Datum references provide a known origin for measurements.

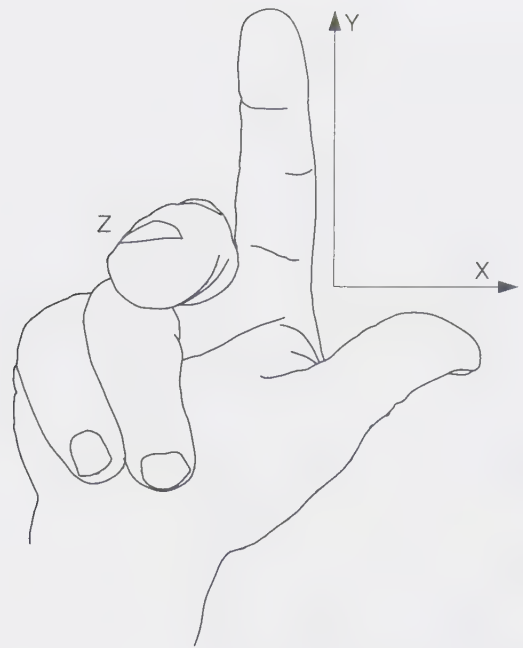
Degrees of Freedom

Datums, according to their form and the specified order of precedence, are used to constrain the *degrees of freedom* for a part. Some tolerances require constraining a limited number of the degrees of freedom and others require constraining all six degrees of freedom. Adequate datums must be used to constrain the appropriate degrees of freedom.

There are three translational and three rotational degrees of freedom. Translational degrees of freedom are along the X, Y, and Z axes. Rotational degrees of freedom are identified as u, v, and w, and they exist around the X, Y, and Z axes. The u rotation is around the X axis, v rotation is around the Y axis, and w rotation is around the Z axis. See **Figure 6-4**.

The relationships of the X, Y, and Z axes are easily remembered using the thumb and two fingers on the right hand. If the thumb and index finger are held at 90° to one another while pressed against a sheet of paper, the thumb pointing to the right will represent the X axis, the index finger pointing to the top of the paper will represent the Y axis, and the middle finger pointing up from the paper will represent the Z axis. See **Figure 6-7**.

The positive direction of rotation around an axis is determined by pointing the right thumb in the direction of the positive axis and wrapping the



Goodheart-Willcox Publisher

Figure 6-7. The directions of the X, Y, and Z axes may be remembered by placing the fingers of your right hand in the position shown.

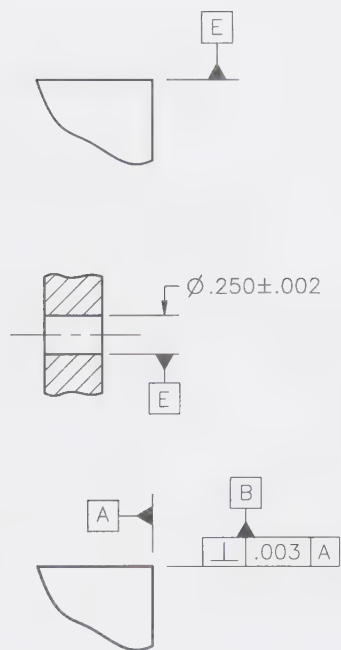
fingers around the axis. The fingers wrap in the direction of the positive rotation.

Datum Identification

Datum features must be identified through the application of datum feature symbols or datum targets. Although standards prior to 1982 permitted datums to be implied, it is no longer acceptable. Even when working to the 1966 or 1973 standard, it is not advisable to use implied datums because of the possible misunderstandings that can occur.

Symbols

A *datum feature symbol* is a square or rectangle connected by a short line to a triangle. The datum feature symbol is used to identify surfaces and features of size as datum features. See **Figure 6-8**. The datum feature symbol typically has a square containing one letter to identify a datum feature. The square may be replaced with a rectangle where two letters are used to identify the datum feature. The square or rectangle is connected by a short leader line to a triangle. The triangle is attached to a part feature, an extension line from a feature, a dimension line, a leader line, or a feature control frame in a manner that indicates the particular feature on the part that is to serve as a datum feature. Datum feature symbols are only applied to physical features.



Goodheart-Willcox Publisher

Figure 6-8. A datum feature symbol may be placed on an extension line, placed in line with a dimension line, combined with a feature control frame, or attached to a feature.

Datum feature symbols are applied to actual features because of the clear definition of requirements that this provides. As previously explained, they are not applied to centerlines on a drawing or in a model because of the confusion that can be caused. Another example of an ambiguous (and wrong) application of a datum feature symbol is to place it on the centerline of a counterbored hole. If the counterbore is not perfectly centered on the hole, then the two features will have separate axes. Such placement does not indicate which feature, the hole or counterbore, establishes the datum.

A **datum target symbol** is used to identify datum targets. See [Figure 6-9](#). The datum target symbol is a circle with a horizontal line across the

center of it. The bottom half of the circle is used to identify the datum target. The top half is left empty except when specifying the size of a datum target area. A size shown in the top half may be specified as a diameter, square, or possibly a rectangle if the proportions of the rectangle are adequately clear.

Flat Surfaces (Planes)

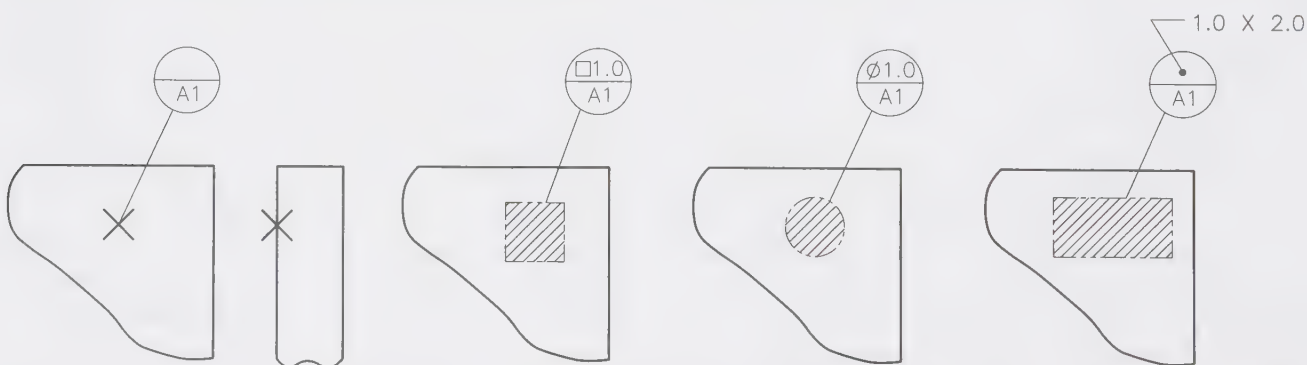
Flat surfaces are commonly used as datum features. See [Figure 6-10](#). A datum feature symbol placed on an extension line from a flat surface identifies that surface as a datum feature. The datum feature symbol may also be attached directly to the surface.

A flat surface identified as a datum feature establishes a datum plane. If the surface of the produced part is not perfect, then the high points on the surface establish the location of the theoretical datum. The theoretical datum may be simulated by placing the part on a very accurate surface, such as a surface plate. A tooling surface or surface plate is normally considered accurate enough to be used as a datum simulator. The datum plane may also be established mathematically by collection data for points on the surface and calculating a plane that passes through the high points. Software for measurement machines typically provides means for doing this.

Cylindrical Features (Axes)

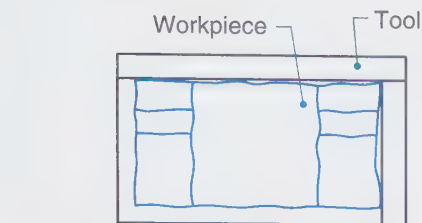
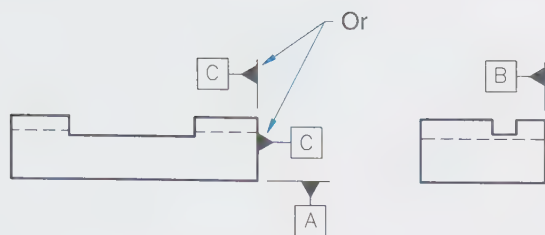
Cylindrical features, such as shafts and holes, are commonly used as datum features. See [Figure 6-11](#). A datum feature symbol placed in line with the diameter dimension for a cylinder defines that cylinder as a datum feature. Attachment to the horizontal segment of a leader that points to the hole is another method that may be used.

When a cylinder is used as a datum feature, the cylindrical feature locates a datum axis. This theoretical datum axis is not to be identified by



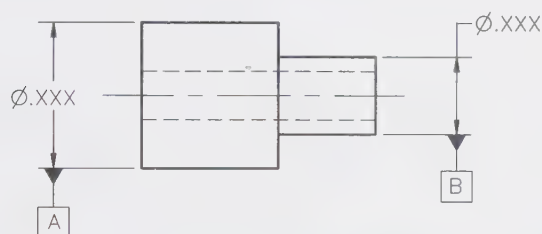
Goodheart-Willcox Publisher

Figure 6-9. The leader on a datum target symbol does not include an arrowhead.



Goodheart-Willcox Publisher

Figure 6-10. Application of a datum feature symbol on either an extension line or a surface has the same meaning.



Goodheart-Willcox Publisher

Figure 6-11. Datum feature symbols are applied to physical features and not to centerlines.

placing a datum feature symbol on the centerline on the drawing. The cylinder must be identified as the datum feature because the cylinder will be used in production and inspection. The centerline cannot be used because it is not a physical feature. As an example, a lathe chuck clamps on a physical feature such as a cylinder; it does not clamp on the centerline. Holding a cylinder in a chuck or collet locates the axis of the cylinder, and thereby locates the datum axis. The centerline is not identified as the datum feature because it cannot be physically contacted.

Targets

It is not always practical to identify an entire feature as a datum feature. A very large feature, such as the outside diameter of a rocket section, is not practical for use as a datum feature. Features subject to distortion, such as castings and

weldments, are also poor features to use in their entirety. Any time features have areas on them that are known to vary significantly, it is usually poor practice to utilize the entire feature to establish a datum. Other examples include surfaces with draft angles or parting lines on them.

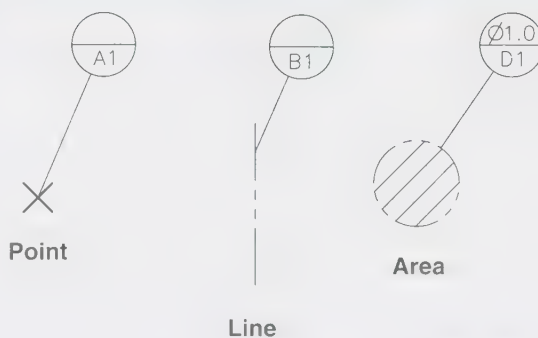
Targets may be used when the entire feature is not used as a datum feature. See [Figure 6-12](#). The types of targets that may be used are points, lines, and areas. Target labels include a letter and a number. The letter identifies the plane or axis. Each target used to establish a single plane or axis is given a number, and the targets for each datum letter are numbered independent of others. If datum plane D is established by three target points, they are labeled D1, D2, and D3.

A leader is extended from the target identification symbol to the target. See [Figure 6-13](#). The leader does not terminate with an arrowhead and it extends radially from the symbol. A horizontal segment is sometimes used in industry but is not standardized. A solid line is used for the leader when the target is on the near side of the feature. A dashed line is used for the leader when the target is on the far side of a drawing view. In CAD models, the leader is always solid. If the target is indicated as movable, as explained later in this chapter, the leader extends from the end of the triangular segment of the symbol.

Pro Tip

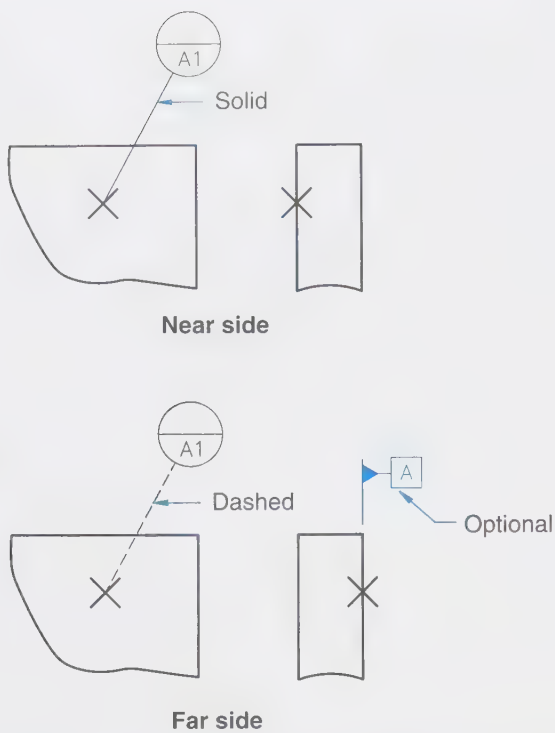
Datum Target Symbol Leader

You are allowed to identify a datum target symbol on the far side by using a dashed leader in a 2D drawing, but it is clearer when views are selected that allow you to use solid leader lines. It is not necessary to create an additional view just to avoid the dashed leader lines.



Goodheart-Willcox Publisher

Figure 6-12. There are three datum target types: point, line, and area.



Goodheart-Willcox Publisher

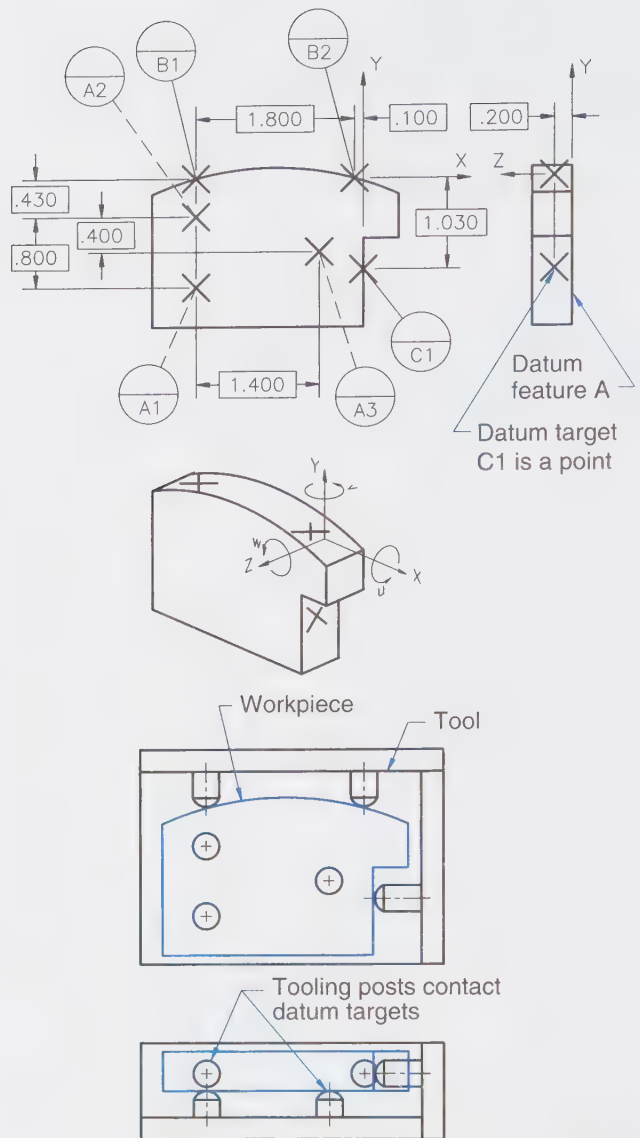
Figure 6-13. A dashed leader from a datum target symbol indicates the target is on the far side of the feature. Dashed leaders are not used when tolerancing solid models.

The datum feature symbol is optional when targets are shown. In the given figure, the datum feature symbol is shown on an extension line from the surface even though targets are also shown. Showing the datum feature symbol on a large feature where targets are far apart may result in misinterpretation, thus causing the entire feature to be used instead of the targets. Typically, the datum feature symbol should not be shown in combination with targets.

Points

Target points on a surface indicate the locations where tooling or processing equipment must attempt to make point contact with the workpiece surface. Surface variations or tooling inaccuracy may result in contact location being imperfect. See [Figure 6-14](#). Target points may be used to establish all the datums required to complete a datum reference frame.

Each target point must have a defined location and be labeled. Points forming a single datum reference frame should be dimensioned relative to one another whenever possible. In the given example, the dimensions that are provided show the relationship between the targets. This allows the targets to be located without the part. A tool or setup can be established without the part being present.



Goodheart-Willcox Publisher

Figure 6-14. Target points may be used to establish all the datums for a part. Target points are commonly picked up by spherical-ended tool posts.

Dimensions used to locate targets may be basic or toleranced dimensions. If target locating dimensions are basic, the ASME Y14.5 standard has in the past indicated that standard toolmaker tolerances apply to the target locations. Some people consider this guideline to be vague because toolmaking standards vary from one company to another.

Although not defined by the current standard, it is possible to define the allowable location tolerance for the target point positions. If and when this is done, a note shall be placed on the drawing to explain what the requirement means. A note is needed because no standard defines the meaning of a tolerance applied to a datum target.

The target point locations shown on a drawing indicate where a production tool or the processing equipment is to make contact to establish the datum reference frame relative to the workpiece. Contacting the datum target points locates the datum planes. When tooling is used to locate the workpiece, the tool must have a feature to pick up each of the identified targets.

Because the targets on the drawing, shown in **Figure 6-14**, are points, point contact must be made. Spherical-ended tool posts are typically used because they result in point contact and are not likely to damage the workpiece.

Standard spherical-ended tool posts and spherical locators are commercially available and should be used. Using standard tooling components is typically less expensive than fabricating special tooling parts.

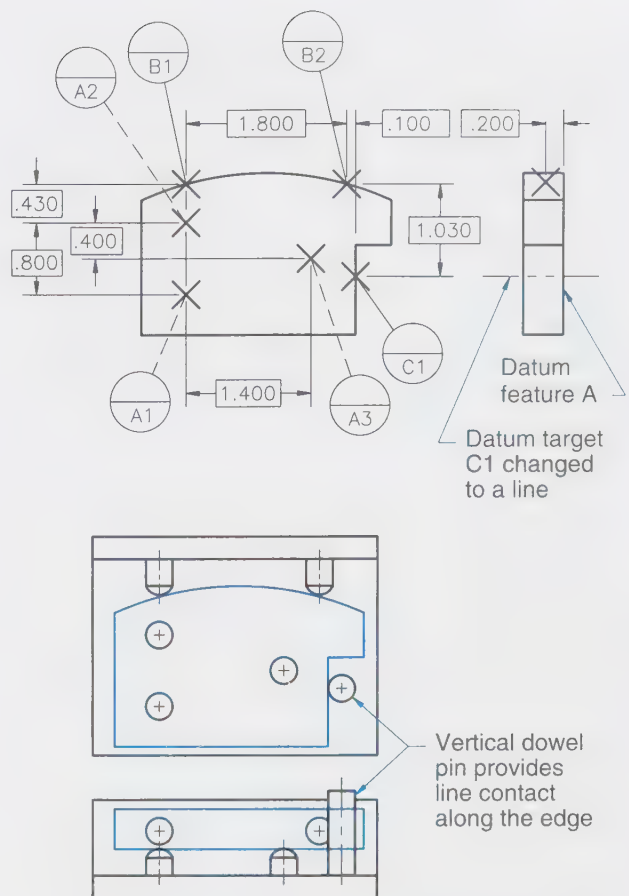
The example in **Figure 6-14** has six targets on the workpiece. Six targets are adequate to completely locate and stabilize the workpiece. They constrain all six degrees of freedom. Targets A1, A2, and A3 establish a plane on which the workpiece rests. This plane constrains z translation and u and v rotation. Targets B1 and B2 constrain y translation and w rotation of the workpiece while on the datum A targets. Target C1 constrains x translation of the workpiece—it stops the part from sliding along the datum B targets.

The three point contact on the primary plane, two point contact on the secondary plane, and one point contact on the tertiary plane is standard when using flat surfaces to establish a datum reference frame and when the part is adequately rigid. This 3, 2, 1 system is also used on irregular surfaces when it is adequate to fully constrain all six degrees of freedom.

No datum feature symbol is required when a datum feature is defined using datum targets. It is allowable to apply the datum feature symbol in addition to the targets. The presence of the datum feature symbol does not indicate that the entire feature is applicable when targets are shown. The targets define the contact locations that must be used to establish the datum.

Lines

Target lines on a surface indicate the locations where the datum target simulators are positioned. The target lines are oriented at the basic dimensions defined on the drawing so that line contact may be made with the workpiece surface. See **Figure 6-15**. The given example is identical to the previous figure except that datum target C1 is now a target line. The datum target line appears as a



Goodheart-Willcox Publisher

Figure 6-15. A datum target line appears like a target point when seen as an end view. Target lines are picked up by tooling features that make line contact.

phantom line in the right side view. The end view of the line, as shown in the front view, looks like a target point. Because the end view of a target line appears the same as a target point, it is essential to look at all views of a drawing before deciding if the targets are points or lines.

Target lines, like target points, must be located and labeled on the drawing. Target lines used to form part of a datum reference frame should be located relative to other datum targets when possible. In the given example, the provided dimensions do show the relationship between the targets.

The tool used for this workpiece is similar to the one in the previous figure, except for the feature that locates datum target C1. Datum target C1 requires a tooling feature that makes line contact. Line contact is achieved in the shown tool by a dowel pin. Contact between the side of the round dowel pin and the flat side of the workpiece results in a line contact.

Line contact along the full length of the target line is only required when the workpiece is perfect. Where the workpiece is not perfect, the dowel

pin will not touch all points along the line. Contact along the full length of the line is not required. It is only required that the dowel simulates the target and be located so that contact can be made along the full length if the workpiece is theoretically perfect.

Comparison of the tools in **Figures 6-14** and **6-15** shows the one in **Figure 6-15** to be simpler in design. This design is also less expensive to produce. When the function of a design allows optional target usage, then target selection should be made in a manner that will lower tooling and product cost.

Areas

Target areas on a workpiece indicate the locations where tooling surfaces that simulate the target area must be positioned to make contact with the workpiece surface. See **Figure 6-16**. The given example shows three surface areas. They are on the bottom of the mounting feet of the part. The three datum target areas define datum A.

Phantom lines are normally drawn around the perimeter of target areas as shown for targets A2 and A3 on the given part. If the target areas are small, the perimeters may be defined by either a continuous line or by using the target point symbol. Whether a phantom line, solid line, or point is used, the specification of target area size is required.

No phantom line is needed for target area A1 on the given part because its perimeter is defined by the geometry of the foot. Datum target A1 is the entire bottom surface on the mounting foot. Its perimeter is therefore shown with object lines.

Target areas are normally located by dimensions, just as is required for target points and lines. An exception is when the areas are defined by the limits of a feature, such as target area A1 in the given example.

In addition to giving dimensions to locate target areas, it is necessary to define the size of the area. Round target areas are dimensioned by placing the diameter inside the top half of the datum target symbol. If the diameter dimension does not fit in the available space, it may be directly applied to the area or placed adjacent to the datum target symbol.

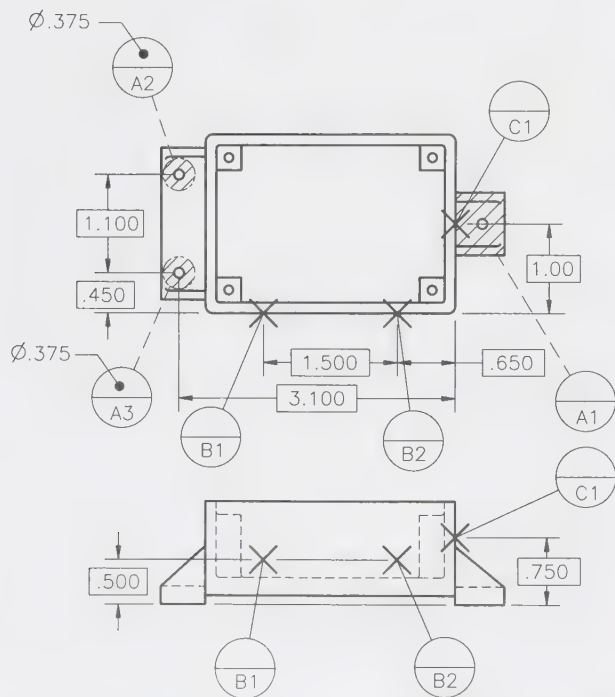
If a target area is square, the square symbol and dimension may be placed in the top half of the datum target symbol. Target areas of any shape other than round or square are sized by a notation placed outside the target symbol or by direct application of the dimensions to the target outline. The size and location must be clearly defined regardless of the target area shape. Size dimensions are not required when supplemental geometry is used to define the target area in a solid model. The supplemental geometry dimensions adequately define the size.

Target areas A1, A2, and A3 in the given example are on the far side of the object; therefore, the leader lines are dashed. As explained earlier, when shown on a 2D drawing, the leader must be dashed to indicate the far side of the object.

Target areas may be any shape or size needed to properly control the design. However, standard tooling components should be considered when selecting the shape and size of the target area. Specifying target areas compatible with standard tooling components will minimize the tool design and tool fabrication costs. As an example, specifying a .500" diameter target area permits the use of a standard tool post. Specifying a .523" diameter target area forces the fabrication of a special tool post.

Establishing Datums and Datum Reference Frames from Datum Features

Datum features on parts are used to establish datums. The datum features may be flat surfaces, features of size, or other feature shapes, such as



Goodheart-Willcox Publisher

Figure 6-16. Datum target areas are outlined with a phantom line and crosshatched.

contoured surfaces. The way in which these features are used to establish datums is dependent on many factors, including the order of precedence and the applicable material boundary.

The datum reference frame created from referenced datum features is established from the true geometric counterparts (TGCs) (theoretical simulators) for the datums, and that is in practice accomplished through some means of physical or mathematical datum simulation. The simulation methods are all imperfect, so a method that is sufficiently accurate for the specified tolerances must be selected.

The ASME Y14.5-2009 standard explains this in terms of using a theoretically perfect datum feature simulator, which is in effect a true geometric counterpart.

Datum Simulation

To simplify explanations of datum reference frames and their application, *datum simulators* will be used to represent TGCs. For purposes of explaining tolerances and datum simulations, the simulators will be shown as though they are perfect.











Simulation in industry, either through tooling or processing equipment, is essential because fabrication and inspection operations must be made in relationship to the specified datums. Simulation may be accomplished with a tool-quality piece of hardware, or it may be done through computer calculations based on data obtained by probing the part with coordinate measuring machines (CMM) or numerical-controlled (NC) machines.

Datum simulation, when done with real rather than theoretical simulators, is a means of approximating the theoretical location of the datums. It is

an approximation because the available simulation methods are imperfect. The simulation methods used should be very precise in relation to the variations on the datum features. The datum simulators used are intended to serve as the TGCs that, in theory, establish the datums from the datum features. The datum feature has variations caused by inaccuracies in fabrication methods. The physical datum simulator is also imperfect but should be much better than the datum feature.

Material Boundary Modifiers

Over many decades, the application of modifiers on datum feature references has advanced. The advances have not reversed past practices but have consistently expanded the ability to define requirements. Prior to 2009, the letter M inside a circle was known only as the maximum material condition symbol. However, it had multiple applications and the requirement communicated by the symbol was in part determined by where the symbol was applied and any tolerances that were applicable to the datum feature. The same symbol is now defined to include all of the past meanings and also expanded to include what is called the *maximum material boundary (MMB)*. See [Figure 6-17](#). The table shown in the figure shows that in the current ASME Y14.5 standard, the letter M inside a circle provides coverage for two items that were not previously included. Those items are the bottom two in the table. That means the MMB symbol can now be applied on datum feature references and includes all the flexibility of previous standards, plus the meaning is expanded to accommodate more feature types as datum features.

Tolerances on datum feature	2009		Pre 2009	
	Symbol / Applicable condition		Symbol / Applicable condition	
Size		MMC		MMC
Size. Form		MMB (Vc)		Vc
Size. Orientation		MMB (Vc)		Vc
Size. Position		MMB (Vc)		Vc
Profile of boundary		MMB		Noted
Profile of surface		MMB		Noted

MMC = Maximum material condition
MMB = Maximum material boundary
Vc = Virtual condition

Goodheart-Willcox Publisher

Figure 6-17. The circle M symbol applied on a datum feature reference means maximum material boundary.

Datum feature references provide the needed information to establish how the simulators are to be used to establish the theoretical datums from the datum features. Based on Rule #2 of ASME Y14.5-2009, when a datum feature reference has no modifying symbol following the datum letter, then the datum feature reference is understood to apply *regardless of the material boundary (RMB)* for the datum feature. When a datum feature reference applies RMB, the simulator is to make contact with the datum feature. Depending on the order of precedence and the geometric tolerances applied to the datum features, the simulator moves to the datum feature by changing one or more of the following: size, orientation, and location. In the example of a flat surface, the simulator moves to the surface. In the example of a hole, the simulator expands within the hole until it makes contact with the hole. The standard uses the term “progresses” instead of “moves” in regard to the datum simulator change that results in contact with the datum feature.

If the datum feature reference is followed by a modifying symbol, either an M or an L inside a circle, then the simulator will be a fixed size. When the letter M is placed within a circle and applied following a datum feature reference, the symbol indicates that the datum is to be established on the basis of the *maximum material boundary (MMB)* of that datum feature. The simulator is fixed in size and, depending on order of precedence, may have a fixed orientation and location.

The requirements created by the RMB, MMB, and LMB modifiers are explained in greater detail in the following sections of this chapter.

Surfaces as Datum Features

Flat surfaces are commonly used as datum features. The means for identifying a flat surface as a datum feature is shown in [Figure 6-10](#). Although continuous flat surfaces do sometimes occur on parts, it is very common for surfaces to have holes and slots in them. These interrupted, noncontinuous flat surfaces are often the features that must serve as datum features.

An interrupted surface may be identified as a datum feature. See [Figure 6-18](#). An interrupted flat surface may be established as a single datum feature by applying a profile tolerance and attaching a datum feature symbol to the feature control frame. The profile feature control frame is shown on an extension line from the surface, but it may also be attached to all the surfaces using three leaders. If interruptions of the surface are very

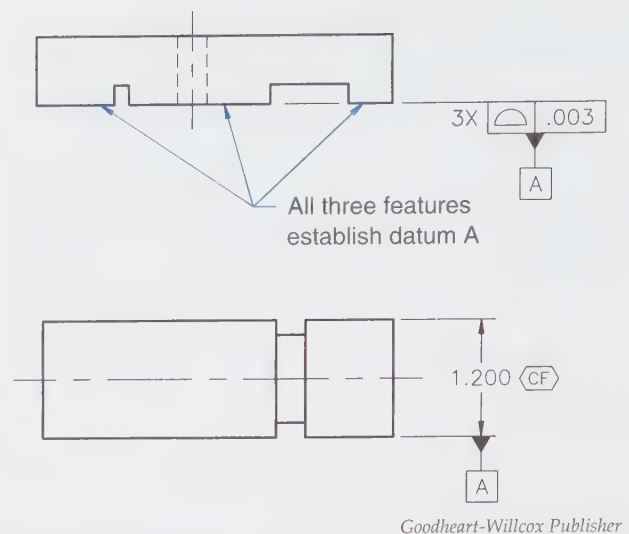


Figure 6-18. Surface interruptions may exist in a datum feature.

large, an extension line may be placed across the interruption to indicate the surfaces are aligned, but the extension line by itself does not indicate a single continuous feature. Past practices did use an extension line in some applications of continuous features, but the current standard no longer supports that practice and it should be avoided. In a solid model, it is acceptable to use the CAD software to associate the three surfaces to the one feature control frame such that selecting the profile tolerance or the datum feature symbol results in the three surfaces being highlighted. Multiple leaders are not used in CAD.

In addition to the 3X notation on the profile tolerance, a note indicating the number of surfaces may also be placed adjacent to the datum feature symbol. The note applicable to [Figure 6-18](#) would state 3 SURFACES or 3X.

The current standard does not specifically show the continuous feature symbol adjacent to a datum feature symbol on a surface, and to avoid ambiguity, such an application should not be used. Application of either a notation of the number of surfaces or the use of a profile tolerance is strongly recommended for continuous surfaces that extend across interruptions so that compliance with the current standard is achieved and there is clarity of intended application. The lower portion of [Figure 6-18](#) shows an acceptable application of the CF (continuous feature) symbol on a feature of size. This application of the CF symbol indicates that the 1.200 diameter forms one boundary of perfect form at MMC that extends the full length of the two cylinders. In regard to the perfect form boundary, the two cylinders are considered as one continuous feature.

Using a surface to establish a datum depends on two things. First, the surface must be identified as a datum feature. Second, it must be referenced in a feature control frame or note. When a datum feature surface is identified, that identification does not by itself indicate how the datum feature is used. It is the reference to the datum feature, in a feature control frame or note, that determines how the datum is used.

Order of Precedence

The order in which datum feature references are made establishes an order of precedence for the datums. Refer back to **Figure 6-3**. The datum feature references are always read from left to right. The first datum is primary, the second is secondary, and the third is tertiary. The primary datum feature does all that its geometry can do in regard to constraining degrees of freedom. The secondary datum feature can only do what the primary datum feature has not already done. It cannot override or change what has already been done by the primary datum feature. Finally, the tertiary datum feature can only do what the primary and secondary datum features have not already done.

Datum features referenced in a feature control frame establish a datum reference frame. The datum reference frame is located and oriented according to the order of precedence specified for the datums. It is important to remember that the datum reference frame is always perfect, regardless of the condition of the datum features.

If two feature control frames both reference the same datums in the same order of precedence with the same material boundary modifiers, then both tolerances are related to the same datum reference frame. If two separate feature control frames refer to different datum features, then those tolerances each establish separate datum reference frames. Even if one or two common datum feature references are shown, one changed datum feature reference creates a different datum reference frame. As an example, two different datum reference frames are created if one feature control frame references datum features A, B, and C, and another references datum features A, B, and D.

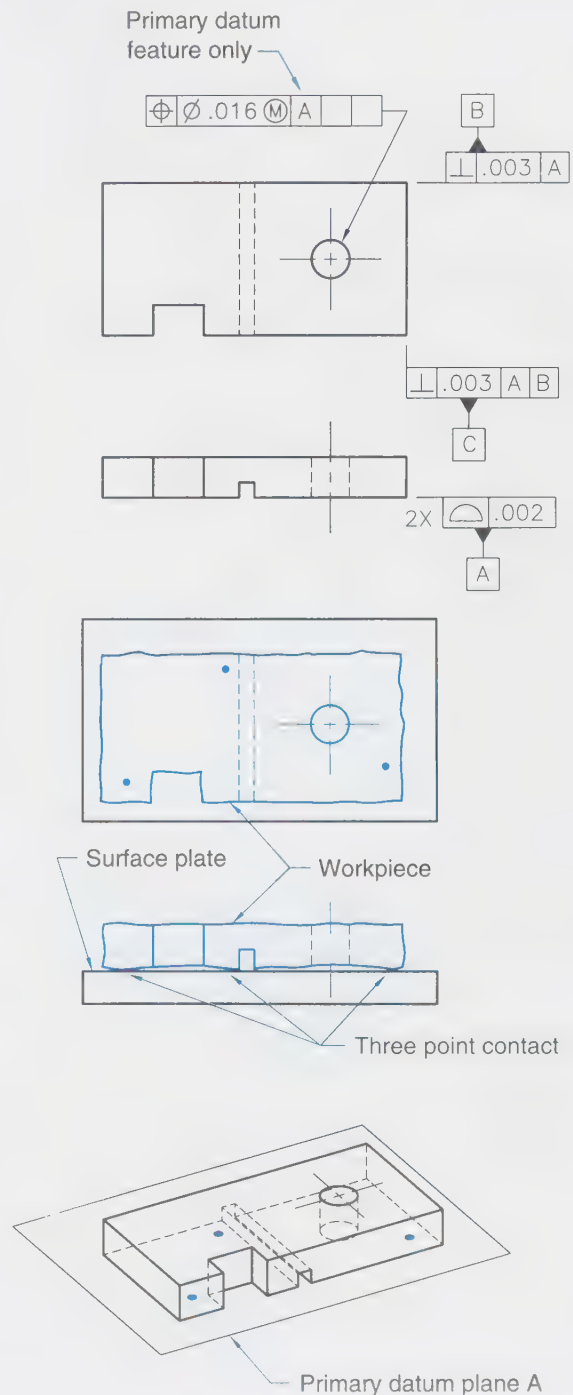
In addition, two datum reference frames are created when the same datum features are referenced in a different order of precedence. As an example, datum features A, B, and C establish a different datum reference frame than datum features B, A, and C.

Primary Datum

The first datum feature reference in a feature control frame is to the *primary datum*. See

Figure 6-19. The primary datum feature establishes the location of the first plane of the datum reference frame if the primary datum feature is a flat surface. Establishing the location of the primary plane is important because the other two planes of the datum reference frame are perpendicular to it.

A position tolerance referencing datum feature A as the primary datum feature is shown in



Goodheart-Willcox Publisher

Figure 6-19. A primary datum plane is located by the primary datum feature on a part.

the given figure. Datum feature A is the bottom surface. A small slot extends across the bottom surface, so a notation of 2X, meaning two surfaces, is included on the profile tolerance to ensure the tolerance and datum feature are understood to be continuous.

The primary datum plane is in theory located by the three highest points on the surface. *Three points define a plane.* The primary datum plane is established by a flat (planar) simulator that contacts the high points on the datum feature. In practice, the simulator may be a flat plate and the workpiece placed on the plate. Assuming a perfect simulator, the workpiece comes to rest on the three high points. The location of the three high points cannot be predicted and may be anywhere on the surface. Therefore, the flat plate (the simulator) must be at least as large as the datum feature. A simulator plate that is smaller than the datum feature may not contact the three highest points.

It is possible that more than three points coincide in the same plane, and more than three points may contact the flat plate. Contact with more than three points is acceptable.

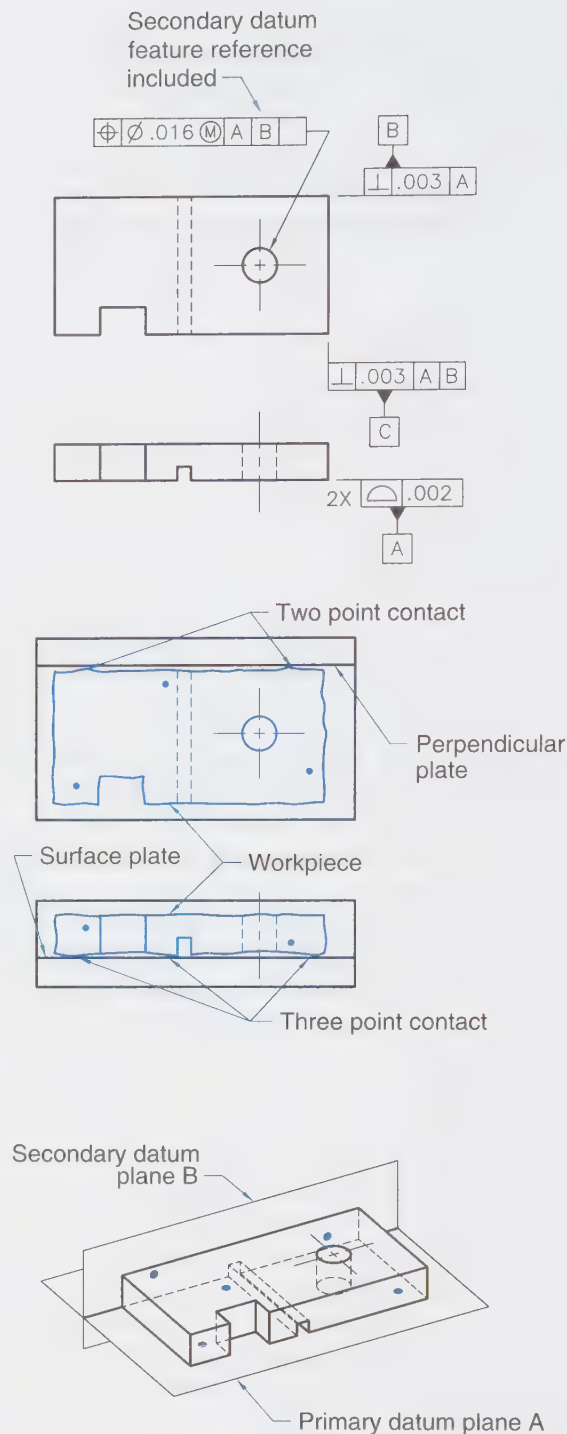
Contact with fewer than three points is not desirable. A convex workpiece may be within tolerance, but would not be stable on the flat plate. If the workpiece is not stable when placed on a flat plate, a means must be used to stabilize the workpiece relative to the datum. In general practice, the part may be shimmed to stop movement. However, a more mathematically specific stabilization procedure is outlined in ASME Y14.5.1. Generally stated, the workpiece must be stabilized on the primary datum plane.

Another means of simulation is to use a machine to probe the surface at many points and take coordinate readings for the points. The readings are used to compute the datum plane location.

When the primary datum feature is a flat surface and the workpiece is resting on the primary plane, three of the six degrees of freedom are constrained. The remaining three degrees of freedom are not constrained by the primary plane. The workpiece is free to translate (slide) and rotate on the primary plane, provided that three point contact is maintained. Secondary and tertiary planes may be used to fully constrain the remaining degrees of freedom and thereby prevent movement of the workpiece on the primary plane.

Secondary Datum

The second datum feature reference in a feature control frame is to the *secondary datum*. See [Figure 6-20](#). The secondary datum feature is used



Goodheart-Willcox Publisher

Figure 6-20. The secondary datum plane is at a 90° angle to the primary datum plane, and is located by the secondary datum feature on the part.

to establish the location of the second plane of the datum reference frame. The secondary plane is perpendicular to the primary plane.

A position tolerance referencing datum B as the secondary datum is shown on the given part. A long edge of the part has a perpendicularity tolerance related to primary datum A, and that

surface is identified as datum feature B. The secondary datum plane must be perpendicular to the primary datum plane and its location established by a datum simulator (a plane) that makes contact with the two highest points on the datum feature. Only two points are needed because the secondary plane is already known to be perpendicular to primary plane A.

The locations of the two highest points are unknown, so the workpiece is pushed against a simulator, such as an angle block or perpendicular plate. Three point contact with the primary plane is maintained. The workpiece will stop when contact is made with the two high points on the secondary simulator, thus constraining two degrees of freedom that were not already constrained by the primary datum. The location of the high points cannot be predicted. Therefore, the simulator must be at least as large as the datum feature.

It is possible that more than two points will contact the plate. Contact with more than two points is acceptable. Contact with fewer than two points is not acceptable. Contacting only one point would permit the workpiece to rotate (rock); therefore, the required rotational degree of freedom for the workpiece would not be constrained. If surface form variation results in only one point making contact, or should the two points be close together, then steps must be taken to stabilize the part.

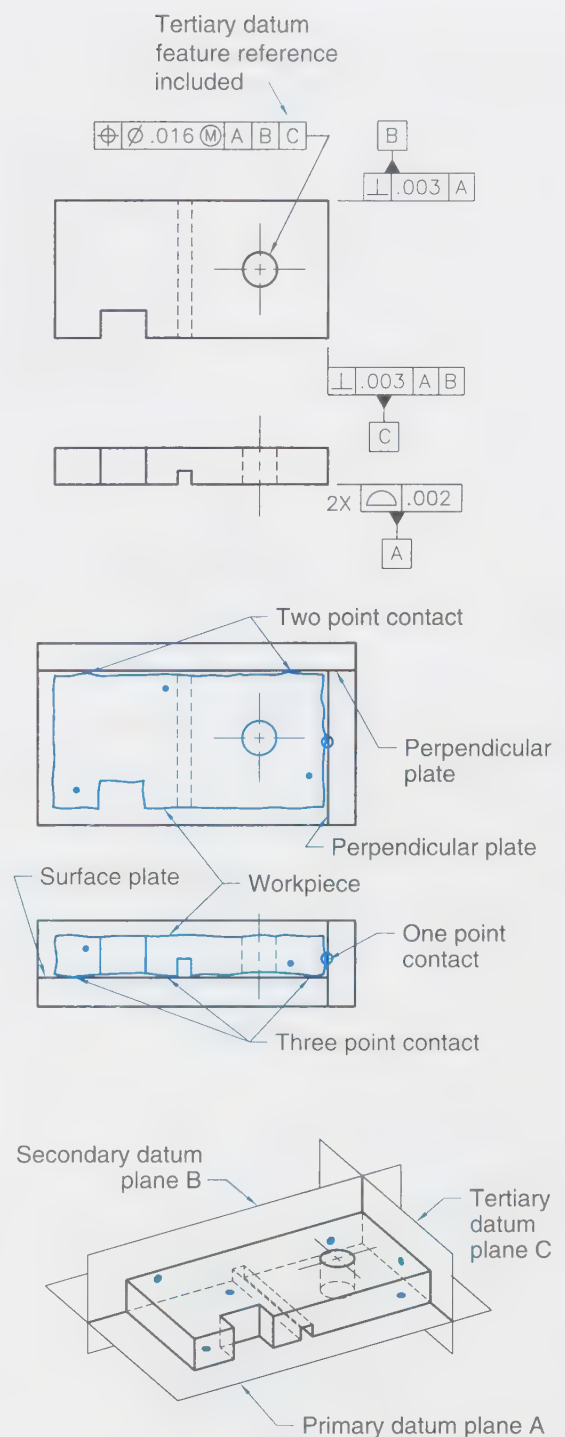
Three rotational and two translational degrees of freedom are constrained when the primary and secondary datum planes are established using flat datum features. The workpiece is still free to translate (slide) along one axis. A tertiary datum is needed to constrain the one remaining translational degree of freedom of the workpiece.

Tertiary Datum

The third datum feature reference in a feature control frame is to the *tertiary datum*. See [Figure 6-21](#). The tertiary datum feature in the given figure is used to establish the location of the third plane of the datum reference frame. The location and orientation of the primary and secondary planes are already established, and the tertiary plane must be perpendicular to both of them.

A position tolerance referencing datum C as the tertiary datum is shown on the given part. The right end of the part has a perpendicularity tolerance related to primary datum A and secondary datum B, and that surface is identified as datum feature C.

The tertiary datum plane must be perpendicular to the primary and secondary planes. The tertiary datum plane is located by one high point on the surface. The tertiary datum plane is perpendicular



Goodheart-Willcox Publisher

Figure 6-21. The tertiary datum plane is perpendicular to the primary and secondary planes, and is located by the tertiary datum feature on the part.

to the primary and secondary planes and its location relative to the datum feature is established by moving a planar datum simulator to make contact with the highest point on the datum feature. Only one point is needed because tertiary plane C only needs to constrain one degree of freedom, the only remaining translation.

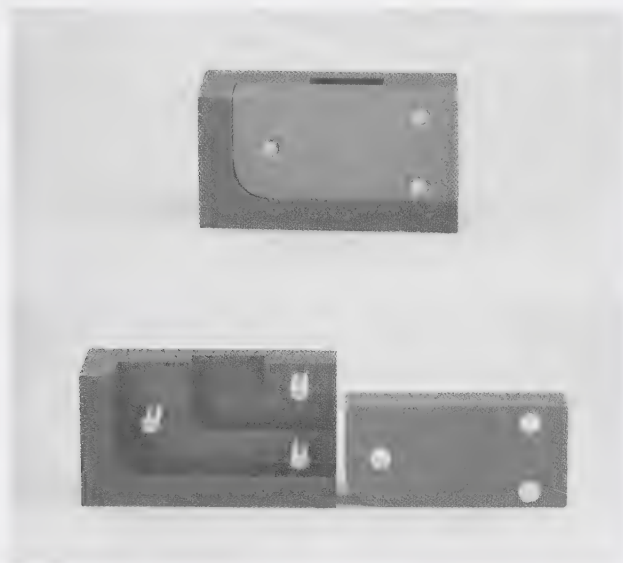
The location of the highest point is unknown, so the workpiece is pushed against a simulator, such as an angle block or perpendicular plate while maintaining three point contact with the primary plane and two point contact with the secondary plane. The workpiece will stop when contact is made with the highest point. The location of the high point cannot be predicted. Therefore, the perpendicular plate must be at least as large as the datum feature.

It is possible that more than one point will contact the plate. Contact with more than one point is acceptable.

All translational and rotational degrees of freedom of the workpiece are constrained when the primary, secondary, and tertiary datums are established. See [Figure 6-22](#). The shown tool simulates three datum features and constrains all degrees of freedom when the workpiece is placed in the tool.

Datum Simulation of Flat Features

Theoretically, a flat datum feature creates a datum plane that coincides with a perfect simulator of the feature. In real-world applications, theoretical datums cannot be contacted, and simulation methods must be used to determine the datum locations with a level of accuracy that is adequate for the workpiece and the tolerances applied to it. Simulation of datum features is required because the parts produced and inspection results must achieve the specified tolerances in relationship to the specified datums. The simulation can be accomplished with a tool-quality piece of hardware, or



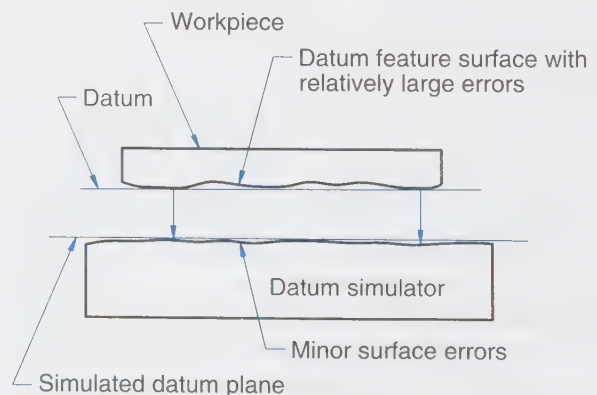
Goodheart-Willcox Publisher

Figure 6-22. The shown tool establishes all three datum planes when the workpiece is inserted into it.

it may be done through computer calculations based on data obtained with coordinate measuring machines (CMM), numerical-controlled (NC) machines, or other measurement equipment.

Datum simulation is a means of approximating, with a high degree of accuracy, the theoretical location of the datums. Datum simulation is meant to locate the datum on the basis of the workpiece datum feature. See [Figure 6-23](#). The datum feature in the figure has variations (shown greatly exaggerated) that are caused by fabrication processes. The simulation tool or process will also have variations, although very small. In the figure, the datum plane is shown in contact with the high points on the workpiece. The simulated datum is in contact with the high points on the datum simulator. When the workpiece is set on the datum simulator, the datum and simulated datum may not be in exactly the same place if the high points on the workpiece do not coincide with the high points on the simulator. Because of the variations in the datum feature and the datum feature simulator, the simulated datum is unlikely to be exactly the same as the theoretical datum.

Variations in a datum simulation tool or process can result in what is typically a small amount of datum simulation error, and that can result in some measurement error for any features measured from the datum. So, there is a theoretical datum plane, and there is a simulated datum plane. The simulated datum plane is the one achieved in production or measurement. The theoretical plane is the one that exists relative to the datum feature without any simulation error. Since tolerances are specified relative to the theoretical datums, any error in the simulation method must be accounted for when measurements are made.



Goodheart-Willcox Publisher

Figure 6-23. Accurate tooling components placed against the datum features on a workpiece simulate the theoretical datums.

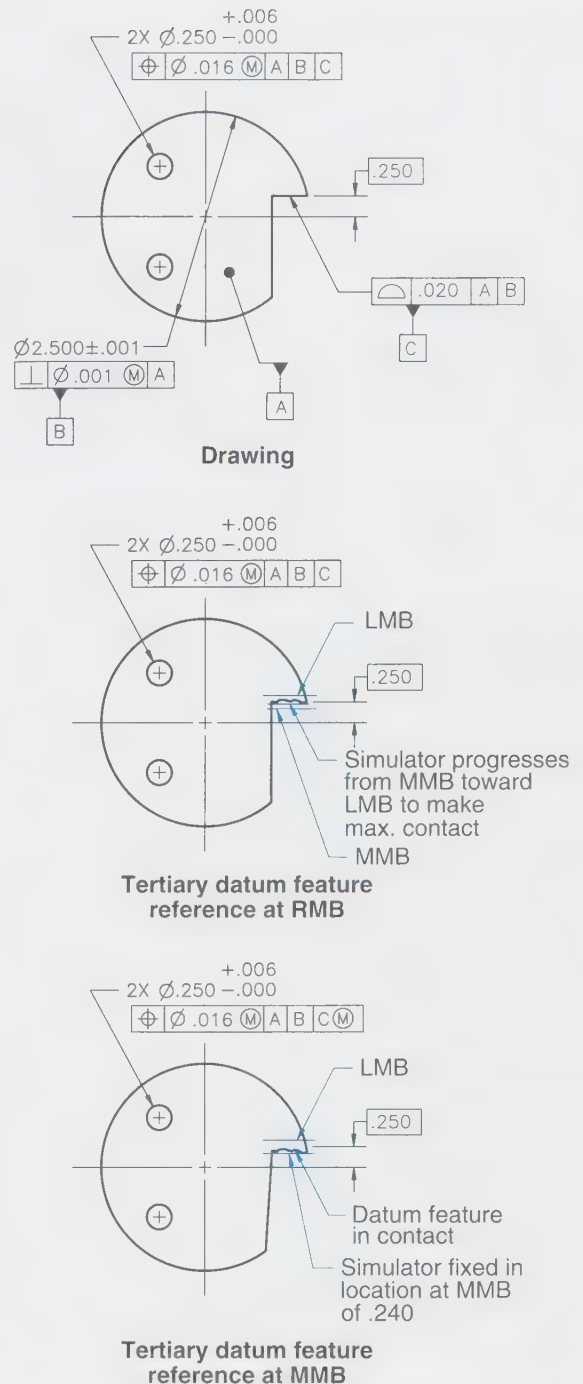
The variations in the datum feature and the simulation method have the potential to cause some error between the actual datum plane and the simulated datum plane established by simulation methods. Different simulation methods may result in a different amount of error between the simulated datum and the theoretical datum. Keeping the simulation error small prevents this from becoming significant.

Datum features on a workpiece are used to locate datum planes. A datum plane is located by the high points on a datum surface (feature). In the given figure, the location of the datum plane is approximated by resting the datum surface on a datum simulator. The datum simulator is often a highly accurate device such as a surface plate or angle block, but there are some variations in the surface. So, there is a small difference between the theoretical and simulated datum. For most situations, the high degree of accuracy in these devices permits measurements from them as if the simulators are the theoretical datum planes. However, if datum simulation is not highly accurate relative to the size of the required tolerances, the simulation error must be considered.

Material Boundary Modifiers on Flat Datum Features

Prior to the approval of ASME Y14.5-2009, material condition modifiers were not used when datum feature references were made to flat datum feature surfaces, because material condition modifiers were only applicable to datum features of size. That has changed. All datum feature references are now assumed to be applicable regardless of material boundary (RMB) and may, when appropriate, be modified with the MMB or LMB symbol. Although a flat surface does not have a maximum or least material condition resulting from size variations, a flat surface may, in some instances, have a maximum material boundary created by the application of profile tolerances. See [Figure 6-24](#). The shown part has a small flat surface identified as datum feature C, and that surface has a profile tolerance applied that is referenced to datum features A primary and B secondary. The profile tolerance creates a maximum material boundary and a least material boundary at the limits of the profile tolerance. This creates a condition where datum feature C may be applicable at RMB, MMB, or LMB. Any tolerance referencing datum feature C defaults to being applicable regardless of material boundary (RMB).

Generally, the MMB and LMB symbol should be avoided when datum features are flat surfaces.



Goodheart-Willcox Publisher

Figure 6-24. A non-size feature may be referenced at RMB, MMB, or LMB when a profile tolerance establishes material boundaries for that feature.

Never use the MMB or LMB modifier on a primary datum feature reference when the feature is a flat surface. Never apply the MMB or LMB modifiers on datum feature references to flat datum features, regardless of order of precedence, unless there are appropriate geometric tolerances on the datum features to establish material boundaries. Generally, the material boundaries must be created

using profile tolerances and those tolerances must be referenced to higher precedence datums.

As stated above, all datum feature references are applicable regardless of material boundary (RMB) unless otherwise specified. If there are no material boundaries for the flat surface, the applicability of RMB has no real impact.

Explanations of datum simulation methods given in the preceding pages were based on the datum feature reference default to applicability at RMB. As previously explained, when the datum feature reference is at RMB, the datum feature is brought into contact with its datum simulator or the datum simulator brought into contact with the datum feature.

The applicability of RMB on a datum feature reference does have an impact if the datum feature has material boundaries. See [Figure 6-24](#). The middle illustration in the figure shows the datum C reference at RMB (no modifier shown). For a flat surface such as datum feature C, material condition boundaries exist. RMB applicability on datum feature C simply means that the datum simulator progresses from the MMB toward the LMB until the simulator makes maximum possible contact with the datum feature.

A flat datum feature that has material condition boundaries may be referenced at RMB, MMB, or LMB, and there are applications where a datum feature reference to a flat surface at maximum material boundary (MMB) is appropriate. When a datum feature reference is made at MMB, the simulator is fixed in location at the MMB. See [Figure 6-24](#). The lower illustration in the figure shows the part with the datum feature C reference at MMB. The datum feature C simulator in the shown illustration is fixed in location at the MMB of the datum feature. Based on ASME Y14.5-2009, the datum feature C must be able to make contact with the datum simulator that is fixed in location at the MMB.

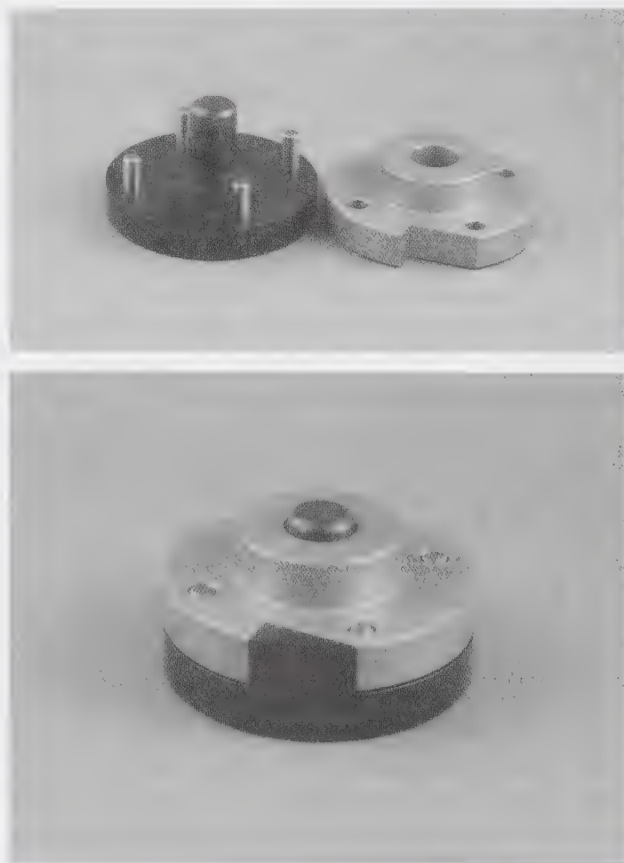
A comparison of RMB and MMB application to a tertiary datum can be seen in [Figure 6-24](#). To achieve design intent, the tertiary datum feature C is meant to stop rotation of the shown part to ensure the two holes are correctly positioned. The tertiary datum feature C has a profile tolerance applied to it and the tolerance zone establishes maximum and minimum material boundaries. When the tertiary datum reference is at RMB, the simulator for the tertiary datum is oriented to the higher precedence datums and it progresses from the MMB toward the LMB to make full contact with the feature. The part may rotate into the position that results in the full contact. The tertiary

datum feature must not move outside its profile tolerance boundary.

When the tertiary datum reference is at MMB, the simulator is fixed in location at the maximum material boundary established by the profile tolerance. When the datum feature referenced at MMB is offset from a higher precedence datum axis as in [Figure 6-24](#), the tertiary datum feature must make at least one point contact with the simulator. The photograph in [Figure 6-25](#) shows datum simulators that are at the MMB locations.

If it is desired for the simulator to be fixed at the nominal location of the tertiary feature, the datum feature reference is followed with the term BASIC or the abbreviation for basic enclosed in brackets [BSC]. This fixes the simulator at the basic location of the feature and the feature is brought into contact with the simulator. It is also permissible to enter a number with brackets to indicate the required location of the simulator. In [Figure 6-24](#), the datum feature C reference would be followed by either [BSC] or [.250]. Further explanation is provided later in this chapter.

The default applicability of RMB on all datum feature references is relatively simple to



Goodheart-Willcox Publisher

Figure 6-25. The shown tool locates the secondary and tertiary datum features at MMB.

accommodate when the datum features are all flat surfaces. **Figures 6-21 and 6-31** both show positional tolerances that include three datum feature references. All three datum features are flat surfaces. No boundary condition modifier is shown on the datum feature references. Therefore, RMB is applicable.

Regardless of the boundary or size of any feature on the part, the surfaces will be placed in contact with flat datum simulators to locate datum planes as shown in **Figures 6-21 and 6-32**. The given part will be placed on the surfaces so that three point contact is made on primary datum feature A, two point contact on secondary datum feature B, and one point contact on tertiary datum feature C.

The three datum planes are mutually perpendicular, even if one of the datum features is made with some orientation or flatness variation. Another way to view this is that each simulator is a plane and each simulator is brought into contact with its associated datum feature while also maintaining the relative orientation of each plane in the datum reference frame.

The three datum planes for the given part are located by the high points on the given surfaces. In **Figure 6-32**, datum plane C is located such that it extends through a portion of the part. This is acceptable because there is no requirement to use the outermost surfaces as datum features.

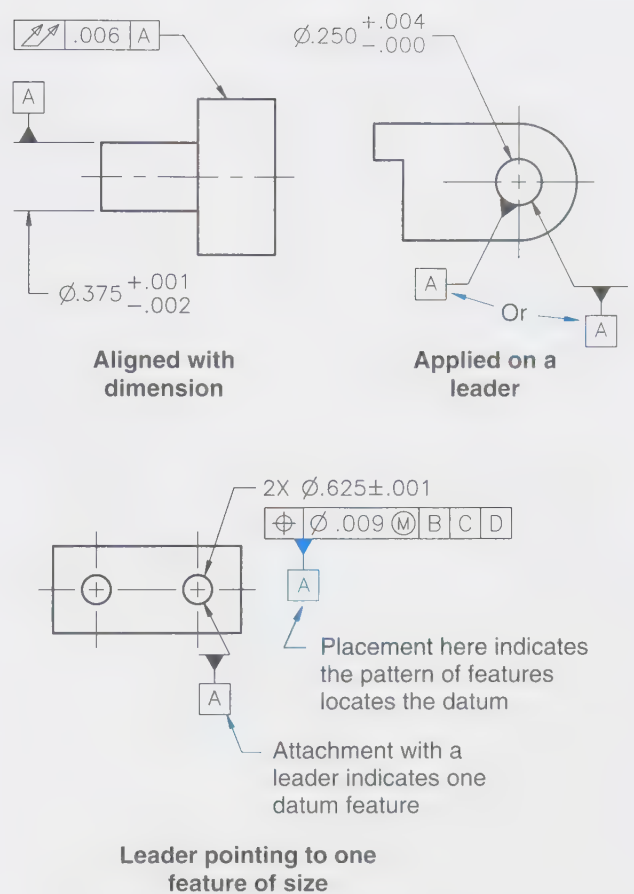
Datum Features of Size

Features of size are often identified as datum features. It is common to use cylindrical features, such as holes and shafts, as datum features of size. Rectangular slots, rails, and tabs are also used as datum features of size.

Placement of the datum feature symbol in line with a size dimension indicates that the feature of size is the datum feature. It is also common practice to attach the datum feature symbol to a feature control frame that is attached to the feature of size. When a datum feature of size is identified, the theoretical datum is typically an axis, centerline, or center plane.

Cylindrical Features

Shafts, holes, counterbores, and other cylindrical items may be identified as datum features of size. See **Figure 6-26**. The datum feature symbol is normally placed in line with the diameter dimension of the cylindrical feature, but it may also be placed on a leader that extends to the circular view of the feature. The datum feature symbol should



Goodheart-Willcox Publisher

Figure 6-26. A datum feature of size is identified by associating the datum feature symbol with the size of the feature.

not be placed on an extension line from the surface when the intent is to identify a datum feature of size. A datum feature symbol on an extension line is used for identifying surfaces as datum features.

The given figure shows three examples of cylindrical datum features. In the first example, the small diameter on a stepped shaft is identified as datum feature A. Datum feature A establishes the datum axis.

The second example shows a hole that is identified as a datum feature of size. The datum identification triangle may be attached to the hole or it may be attached to a leader that points to the hole. Both methods indicate that the hole establishes a datum axis.

The third example shows a part with two holes. A datum feature symbol is connected to one hole with a leader. This identifies only one hole as a datum feature. Placement of the datum feature symbol on the leader of the callout for the two holes or on the feature control frame for those two holes would establish the hole pattern (using all holes in the pattern, not just one) as the datum feature.

Rectangular Features of Size (Parallel Surfaces)

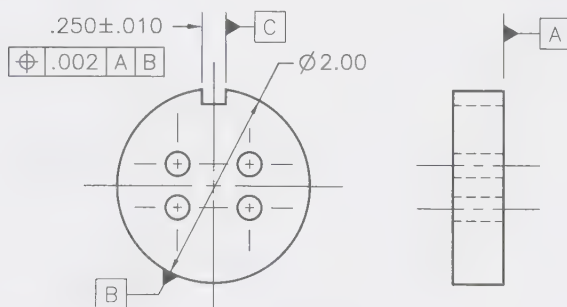
Slots, rails, and tabs are often used as datum features of size. See **Figure 6-27**. The given figure shows a keyseat that is position tolerated relative to datum A primary and datum B secondary. The width of a keyseat is identified as a datum feature of size, datum feature C. Datum C is a plane that is perpendicular to primary datum A, is centered on datum B, and passes through the center of the keyseat. The means of locating datum C is explained in the following sections and is dependent on the order of precedence, the applicable material boundary modifier, and whether or not the translation symbol is shown.

The datum feature symbol is placed in line with the dimension. It is incorrect to show the datum feature symbol on the centerline. Placement of the datum feature symbol on an extension line from one side of the keyseat would not have the same meaning as its placement on the dimension. Placement on an extension line would indicate a single surface as a datum feature.

To identify a rectangular feature of size as a datum feature, the datum feature symbol may be in line with or on the dimension line. It may also be attached to a feature control frame that is applied to the feature.

Datum Simulation

In theory, a datum feature of size referenced RMB is simulated by something called an actual mating envelope. For a primary datum, the actual mating envelope is free to float (not constrained). The actual mating envelope for a primary datum feature is the perfect geometric shape that is collapsed (contracted) around an external datum feature or expanded within an internal datum feature. For a secondary or tertiary datum feature



Goodheart-Willcox Publisher

Figure 6-27. A center plane is identified as a datum by placing the datum feature symbol in line with the feature width dimension.

of size, the actual mating envelope is related (constrained) to applicable higher precedence datums.

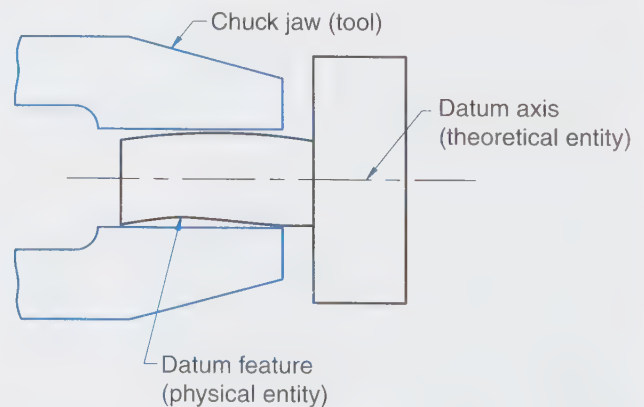
When the datum references are applicable at MMB or LMB, the theoretical simulation is similar to the explanation for RMB, except the simulator is fixed in size to the appropriate material boundary. More about the simulation methods will be explained in following pages.

Figure 6-28 provides a conceptual look at a practical way in which a datum axis may be established from a shaft that is referenced as a primary datum feature of size RMB. In this example, the shaft is placed in a machine chuck to establish the axis location. Of course, a machine chuck is not a perfect simulator because it does not completely enclose the shaft as does a theoretical actual mating envelope. Another option is to clamp the shaft in a collet. The collet may contact more of the surface, but it is not a perfect representation of the actual mating envelope. In both examples, the axis of rotation for the clamping device acts as the simulated datum axis. Each of the simulation methods is intended to make sufficient contact with the surface of the workpiece to adequately establish the datum axis.

Tolerance requirements on the workpiece must be evaluated to determine the needed level of accuracy for datum simulation equipment. Although a lathe chuck or collet is not perfect, their accuracy is sufficient for many datum simulation needs.

Datum Feature References in a Feature Control Frame

Identification of a datum feature has no impact on fabrication requirements unless a reference is made to the datum. Generally, datum feature



Goodheart-Willcox Publisher

Figure 6-28. A datum feature of size is simulated by a tooling device that picks up the feature of size.

symbols should not be placed on a drawing unless references to the datum features are made. Datum feature references may appear in feature control frames or in notes.

The number of datum feature references shown in a feature control frame depends on the type of tolerance specification and the level of control being specified. Some tolerance types, such as parallelism, may need only one datum feature reference. Position tolerances generally require more than one datum feature reference. Regardless of the tolerance type, the number and selection of datum feature references is determined on the basis of the degrees of freedom that need to be constrained. Datum feature reference guidelines provide flexibility for identifying the needed number and type of datums.

Selection of the appropriate number of datums is one step in completing the datum feature reference portion of a feature control frame. The order in which to reference them must also be determined because the order of precedence impacts how the part is located in the datum reference frame.

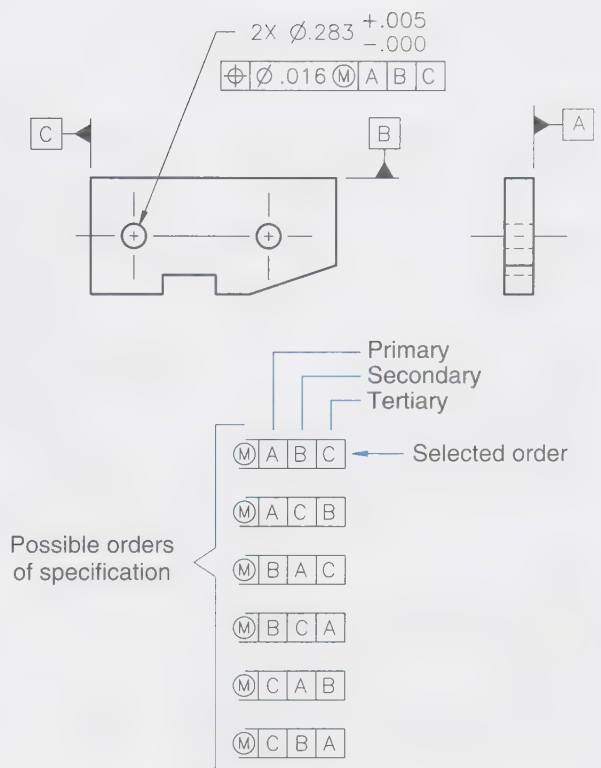
Order of Precedence

If a feature control frame includes references to three planar datum features (A primary, B secondary, and C tertiary), then a single unique datum reference frame is created. Any tolerance specification that references datum features A, B, and C in the same order of precedence will be measured relative to the same datum reference frame.

Any tolerance specification that changes the order of precedence for the datums results in a different datum reference frame. See [Figure 6-29](#). By varying the order of three datum feature references, it is possible to create six datum reference frames. Assuming the correct three datum features are selected, only one of the six possible datum reference frames is optimum for a specific tolerance.

Care must be taken to select the correct datum features and to reference them in the most appropriate order of precedence. Datum selection and the order of precedence are determined on the basis of design function and manufacturing considerations.

The given figure shows datum features A, B, and C on a relatively simple part. The design function requires that datum feature A sit flat on a mating part when two bolts are passed through the holes. Making datum A primary ensures that at least three points will be in contact with the mating part. Using this surface as primary also makes sense for fabrication and inspection of the two holes.



Goodheart-Willcox Publisher

Figure 6-29. The order of precedence for datum feature references affects the requirements shown in the feature control frame.

Edge faces on the part are identified as secondary and tertiary datum features. Datum feature B is selected to be secondary because it is the longest surface and will ensure alignment of parts in the assembly. This surface is also better than datum feature C for stabilizing the part in fabrication. Selection of datum feature A as primary, and B as secondary, results in datum feature C being the tertiary reference.

The best order of datum precedence for this part is A primary, B secondary, and C tertiary. There are five alternate orders of precedence that could be used, but none of them meet the design or manufacturing needs.

Based on the selection of datum A as primary, datum B as secondary, and datum C as tertiary, the workpiece can be located in a tool or machine setup as shown in [Figure 6-30](#). Three mutually perpendicular datum simulators are set up for locating the workpiece. Primary datum A must make contact on at least three points. Secondary datum B must make contact on at least two points. Tertiary datum C must make contact on at least one point.

A physical tool is not required to establish the datum reference frame for a part. The datum reference frame may be mathematically established utilizing point data measured on the datum features.

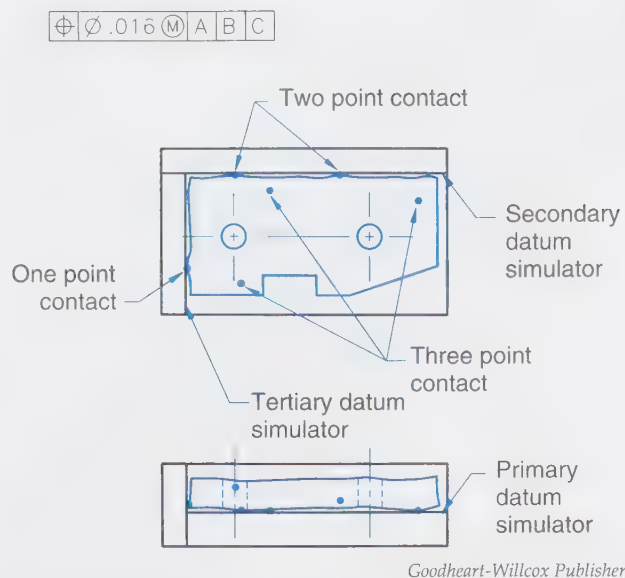


Figure 6-30. The specified order of precedence sets the requirements for how the workpiece must be located in the datum reference frame.

Number of Datum Feature References

If the datum features on a part are flat features and a three-plane datum reference frame is desired, then it is necessary to reference three datums. See **Figure 6-31**. Proper specification of a position tolerance for the holes in the given part requires that three datum feature references be shown. Three surfaces on the part are selected to serve as datum features. The selection of surfaces is made through the process already described. References to the datums are inserted in the position tolerance specification. Datum simulation for the referenced datums is shown in **Figure 6-32**.

If one of the datum features is a feature of size, then it is not always necessary to reference three datums to establish the three planes in the datum reference frame. See **Figure 6-33**. The position

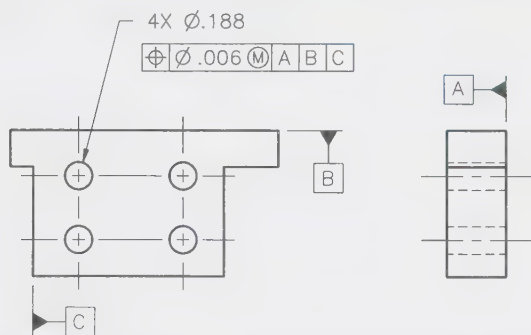


Figure 6-31. Three datum surfaces may be used to establish a datum reference frame.

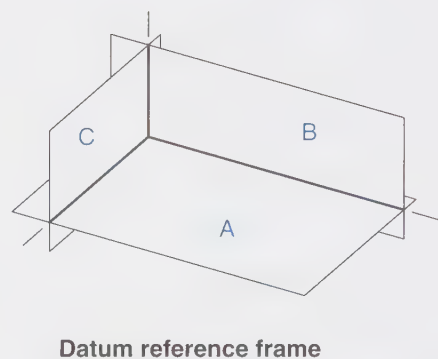
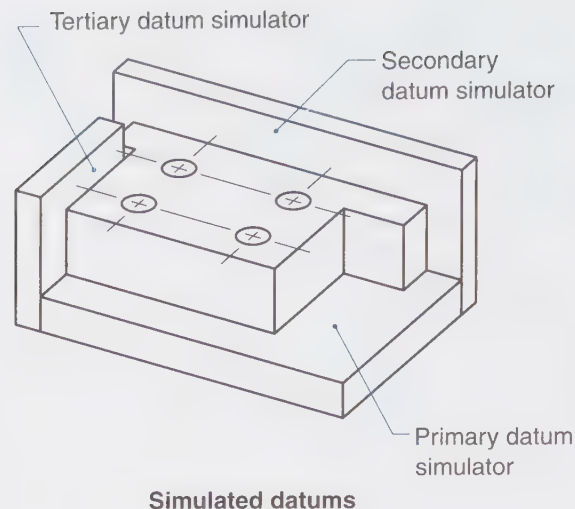


Figure 6-32. Three flat tooling surfaces may be used to simulate the datum reference frame that is created by three datum feature surfaces.

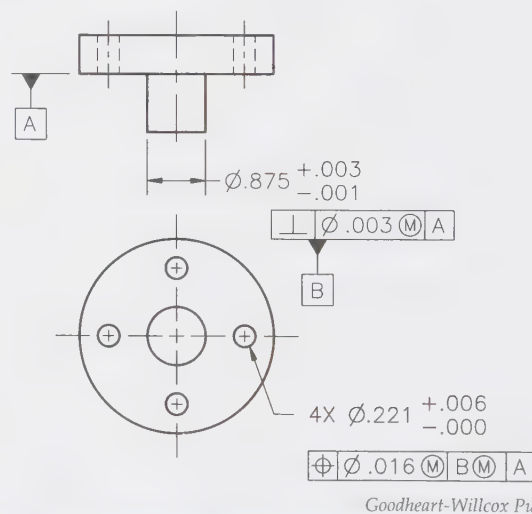


Figure 6-33. One datum feature of size and one datum surface may be used to establish a datum reference frame.

tolerance for the four holes references two datum features. The primary datum feature reference is to a pilot shaft. Referencing datum feature B as the primary datum establishes datum axis B and two intersecting planes at the axis. Referencing a second datum feature, which is a flat surface perpendicular to the axis, establishes the third plane in the datum reference frame. Thus, the axis and plane as used in this example establish all three planes of the datum reference frame.

Figure 6-34 shows a part on which one surface and two features of size are identified and referenced as datum features. In this example, the primary datum feature is a surface. It locates the primary datum plane of the datum reference frame.

The secondary datum feature is a hole. The second and third planes of the datum reference frame are located on the datum axis created by the hole. The secondary datum axis and the associated planes are perpendicular to the primary plane.

The tertiary datum feature is needed to prevent rotation of the planes that pass through the secondary datum axis.

The number of references shown in a feature control frame is affected by the type of datum features and the order of precedence. The required number of references is also affected by the type of tolerance being specified, and the desired level of control.

Material Boundary Modifiers on Datum Features of Size

Material boundary modifiers for regardless of material boundary (RMB), maximum material boundary (MMB), and least material boundary

(LMB) are applicable to datum feature references. The following explains how the material condition boundaries are applied and how they are simulated.

Datum Feature References RMB

Based on Rule #2 of ASME Y14.5-2009, all datum feature references, for features of size and surfaces, are applicable regardless of material boundary (RMB) unless otherwise specified. For a datum reference to a feature of size, RMB means that the simulator (the actual mating envelope) shall make contact with the datum feature. The process utilized to establish contact and the extent of contact will vary depending on the geometry of the datum feature, the order of precedence of the datum feature reference, and tolerances that are applied to the datum feature.

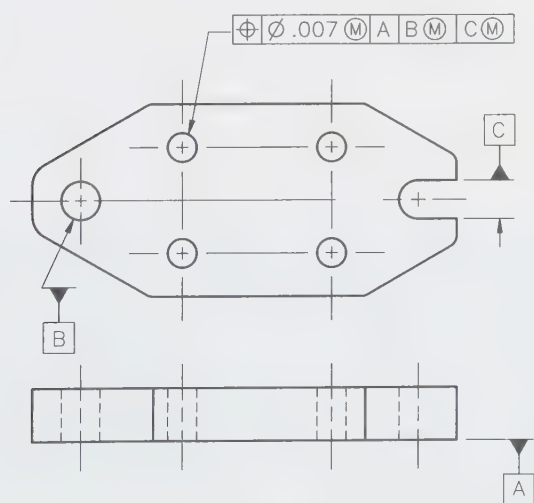
Datum feature references may also be modified with the maximum material boundary (MMB) or the least material boundary (LMB) symbol. When a datum feature reference is modified with the MMB or LMB symbol, the datum simulator has a fixed size. The order of datum precedence and applicable tolerances have an impact on the size and any orientation or location requirements for the simulator. See **Figure 6-17**.

Datum Feature References MMB

Features of size may have multiple boundaries that are created by the tolerances that are applied. The size limit for a feature that contains the maximum amount of material is known as the maximum material condition (MMC) and is within this textbook referred to as either the MMC (which includes form considerations) or the maximum material size (which is only the size limit).

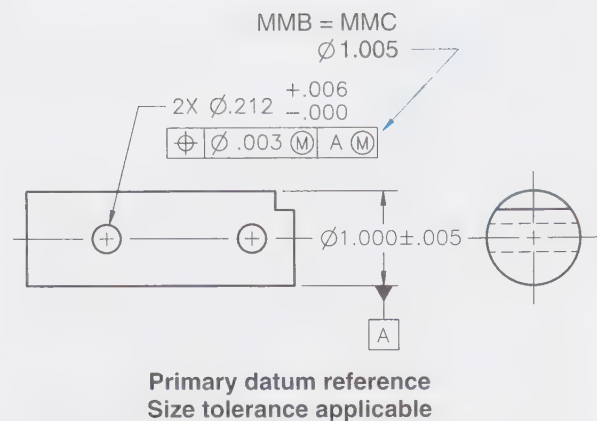
When only a size tolerance is applied to a feature, the MMC for the feature is also the maximum material boundary (MMB) for that feature. The MMC and MMB are coincident. See **Figure 6-35**. The position tolerance on the two holes reference the shaft diameter as primary datum A and the MMB modifier is included. The shaft diameter is identified as datum feature A and only has a size tolerance applied. The MMB of the datum feature is equal to the 1.005" diameter MMC of the shaft because no other tolerances are applied.

If the feature of size has a straightness tolerance applied to define the allowable variation of the derived median line, then the MMC and form tolerance combine to create a maximum material boundary. See **Figure 6-36**. The same part from **Figure 6-35** is shown, except a straightness tolerance is applied to the feature of size. The straightness tolerance applied to a feature of size combines with the MMC size to create a virtual condition.



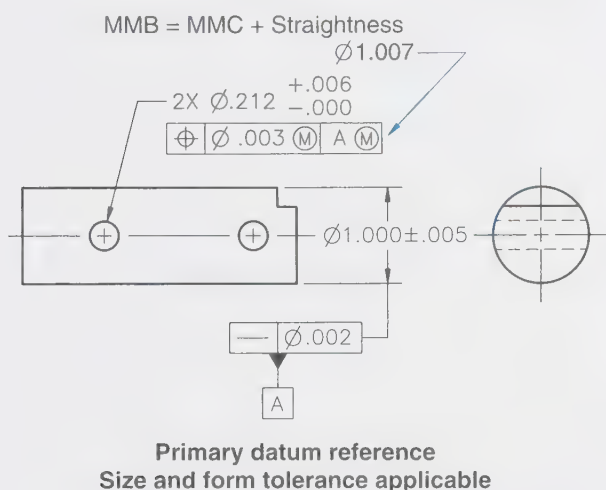
Goodheart-Willcox Publisher

Figure 6-34. One datum feature surface and two datum features of size may be used to establish a datum reference frame.



Goodheart-Willcox Publisher

Figure 6-35. A primary datum reference at MMB requires the simulator to be equal to the maximum material size if Rule #1 has not been superseded by note or form tolerance.



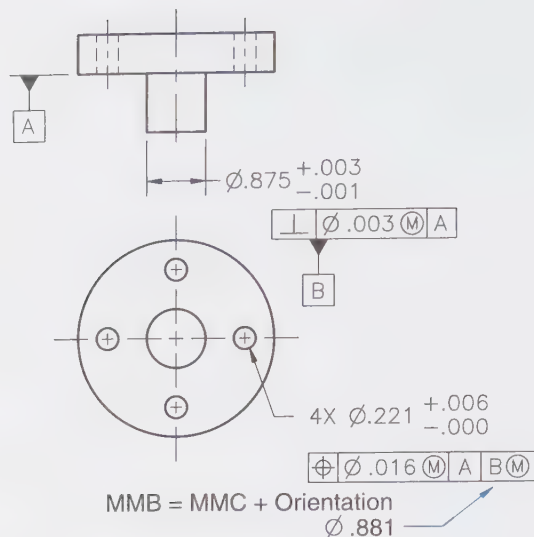
Goodheart-Willcox Publisher

Figure 6-36. A primary datum reference at MMB requires the simulator to be equal to the maximum material boundary when a form tolerance is applied in a manner that supersedes Rule #1.

That virtual condition in this example is an MMB of 1.007" diameter.

The applicable MMB size is affected by the order of precedence of the datum feature reference as well as the tolerances applied to the datum feature. If an orientation tolerance is applied to a secondary datum feature of size, then the MMC and orientation tolerance combine to create another MMB. See [Figure 6-37](#). The secondary datum feature has a size tolerance and an orientation tolerance. The MMC size and orientation tolerance combine to create an applicable MMB of .881" diameter.

If a datum feature of size includes a size, form, orientation, and positional tolerance, the applicable MMB must be determined using the tolerance that is applied relative to higher order precedence



Goodheart-Willcox Publisher

Figure 6-37. A secondary datum reference at MMB results in a simulator sized to the virtual condition of the feature relative to any higher order precedence datums.

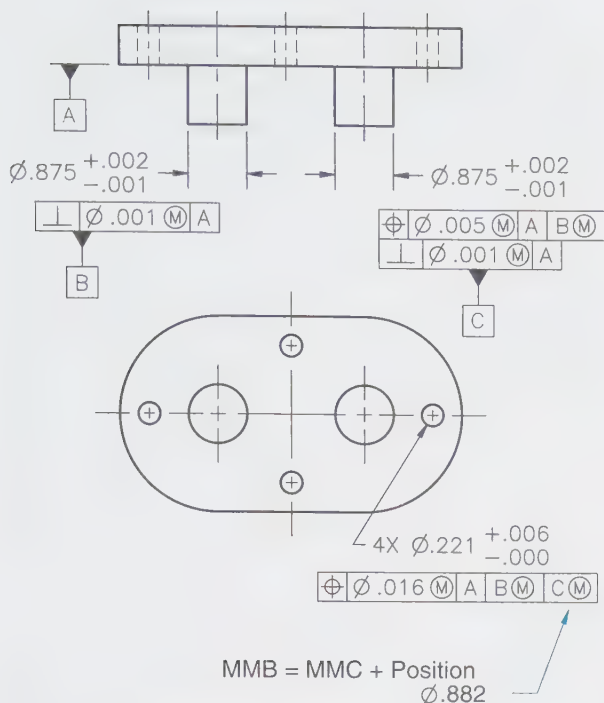
datums. The applicable boundary is always determined in the context of the datum feature reference and its order of precedence. See [Figure 6-38](#).

In the figure, there is a position tolerance applied on four holes and that tolerance references a tertiary datum feature C at MMB. The size of the tertiary datum simulator must be determined. The tertiary datum feature has a size tolerance, perpendicularity tolerance, and position tolerance. The tertiary datum feature has a position tolerance that references a higher precedence datum. Therefore, the MMC and location tolerance combine to create a maximum material boundary. The correct MMB for the tertiary datum simulator is calculated using the MMC size plus the position tolerance. The applicable MMB is equal to .882" diameter and that is the size for a simulator based on the tolerances shown in the figure.

Pro Tip

Applicable MMB

Showing the translation modifier on the tertiary datum reference in [Figure 6-38](#) would indicate the tertiary datum simulator is free to translate. That removes the position tolerance from the calculation of the applicable MMB, because the simulator is free to translate to the location of the datum feature. In this case, only the orientation tolerance is added to the MMC size when calculating the applicable MMB.



Tertiary datum reference
Size and position tolerance applicable

Goodheart-Willcox Publisher

Figure 6-38. A tertiary datum reference at MMB results in a simulator sized to the virtual condition of the feature relative to any higher order precedence datums.

The MMB is outside the material of the part whenever there is an applicable geometric tolerance. The MMB is equal to the MMC only when there is no applicable tolerance other than size tolerance.

Datum Feature References LMB

Regular features of size, irregular features of size, and surfaces may have multiple boundaries that are created by size and geometric tolerances. In the following paragraphs, the applicable material boundary for a datum feature reference that includes an LMB modifier is explained.

For a regular feature of size, the size limit or tolerance boundary that contains the least amount of material is known as the **least material condition (LMC)**. That boundary is impacted by any applicable tolerance. If there are no geometric tolerances applied, then the LMC is equal to the **least material size (LMS)** (which is only affected by the size tolerance). When a datum reference is to a regular feature of size without geometric tolerances and that reference includes the least material boundary modifier, the applicable boundary is equal to the LMC.

When an irregular feature of size or a surface has a profile tolerance applied, two material

boundaries are created. The boundary resulting in the least material is known as the least material boundary.

A datum feature reference that includes the LMB modifier is applicable to the **least material boundary (LMB)**. For a regular feature of size without geometric tolerances, the LMB is equal to the LMC. For an irregular feature of size or surface, the LMB is the profile boundary that contains the least material.

The LMB is always inside the material of the part except when the produced feature is at the least material size. Calculations to determine LMB are similar to those for MMB, except the tolerances are applied on an LMC basis and the datum simulation boundaries are established by determining the applicable boundary that is within the material.

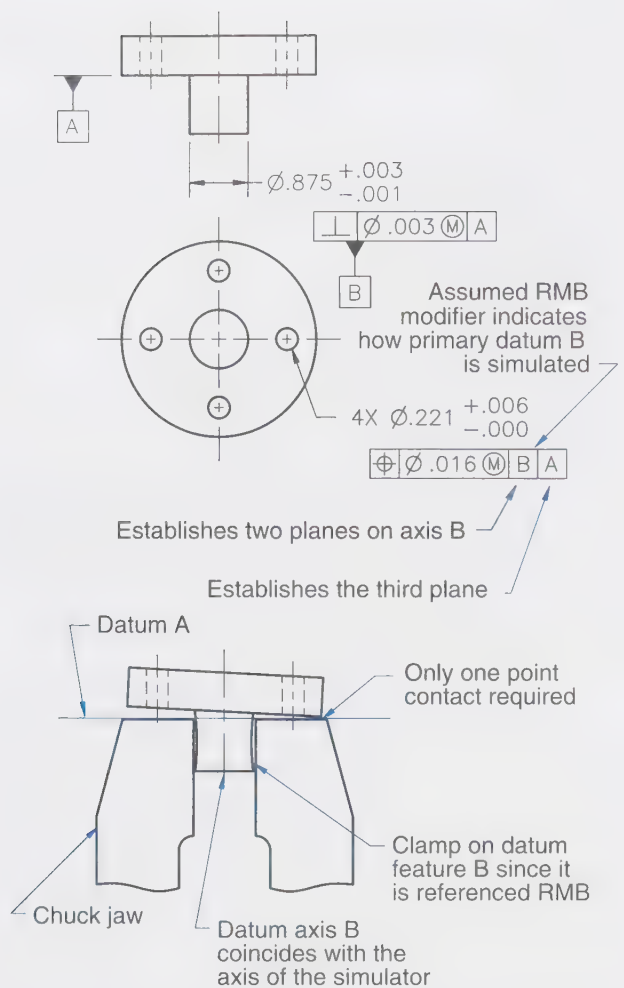
Further Considerations of Modifiers on Datum Feature References

Careful attention is needed for the selection of the appropriate material boundary modifier for each reference to a datum feature of size. Generally, the application of MMB on a datum feature reference can reduce detail part cost, because it increases allowable variation. However, the application of the MMB modifier does not always reduce assembly cost because the additional detail part variation may make assembly difficult. The application of MMB on a datum feature reference may also adversely affect function because of the additional allowable variation. The MMB modifier should be used on datum feature references only when design function and assembly requirements can tolerate the additional variation that is allowable as a result of the MMB modifier.

RMB on a Primary Datum Feature Reference

Placing a datum feature symbol on a diameter dimension establishes a feature of size as a datum feature. See [Figure 6-39](#). The shaft on the given part is identified as datum feature B, and it is referenced as the primary datum feature in a position tolerance. Datum feature B provides a physical feature from which the location of theoretical datum axis B can be determined.

The given figure shows one surface of the flange as datum feature A. The shaft that acts as datum feature B has a perpendicularity tolerance of .003" diameter relative to datum A, with the tolerance applicable at MMC. No material boundary modifier is shown on the datum feature reference, so RMB is applicable to the datum feature A reference.



Goodheart-Willcox Publisher

Figure 6-39. A primary datum feature referenced at RMB requires the datum to be simulated by contacting the surface of the datum feature.

The symbol for datum feature B is attached to the feature control frame that shows the perpendicularity requirement. Showing the datum feature symbol in this manner indicates the same thing as if the datum feature symbol were applied to the size dimension as in **Figure 6-26**.

Four holes in **Figure 6-39** have a position tolerance of .016" diameter at MMC relative to primary datum feature B at RMB and secondary datum feature A. RMB is assumed on datum feature reference B, because no modifying symbol is shown. Referencing datum feature B as primary at RMB has a specific effect.

Datum axis B is located by contracting a cylinder (an actual mating envelope) around datum feature B. Because datum B is primary, the datum axis locates two of the planes in the datum reference frame. Because datum B is referenced at RMB, the datum axis must be located by a simulator that

makes the maximum possible contact with datum feature B. This is typically simulated by a device that clamps on the surface of the shaft. A machine chuck (or collet) may be used to simulate the datum axis. The jaws of the chuck are adjusted to make contact with the shaft regardless of its size or form (RMB). The datum axis will coincide with the axis of the chuck when the shaft is clamped in place.

Hole locations on the shown part may be inspected with the shaft clamped in a chuck, provided the secondary datum is also located. To locate the secondary datum, the shaft would be moved into the chuck far enough for secondary datum feature A to make contact. Point contact with datum feature A might be all that is possible because the shaft and flange may not be perfectly perpendicular.

Changing the order of datum precedence or the applicable material boundary modifier would establish a different requirement. It should be noted that in this example datum feature B has an orientation tolerance relative to datum feature A, and the holes have a position tolerance that references B primary and A secondary. In the given example, datum A is used as a primary datum feature for one tolerance and a secondary datum feature for another tolerance. This is allowed, but care must be taken to ensure the applicable MMB may be calculated if datum feature references are made at MMB.

Pro Tip

Theoretical and Practical Application

The simulation method in the above example is a practical application and not meant to be theoretically perfect. To be theoretically perfect, the chuck would completely enclose all of the datum feature B surface and the end of the chuck would be a surface that could potentially contact all points on the datum feature A surface.

It is important to realize that secondary datum plane A is perpendicular to primary datum axis B. This is true even if datum feature A is not perpendicular to the shaft. The datum planes must be perpendicular because a datum reference frame is always made of perpendicular planes that define a coordinate system.

Simulating datum axis B on the given part constrains two rotational degrees of freedom and constrains two translational degrees of freedom. Two of the planes in the coordinate system intersect on datum axis B, and the third plane is perpendicular

to the axis. The three planes are mutually perpendicular. In simpler words, the datum axis locates two of the planes and establishes the orientation of all three planes of the coordinate system. Datum feature A only constrains translation of (locates) the third plane.

The given figure shows only one of the several possible position tolerance specifications that could be provided for the four holes. Selecting the proper specification requires consideration of the application for the part. The primary reference to datum feature B at RMB is appropriate for a situation where the shaft presses into a hole. RMB is applicable for a press fit between parts, because regardless of the feature sizes, the axes of a hole and shaft will coincide when the parts are pressed together.

A practical application of the above explanation is shown in **Figure 6-40**. The tool is a collet and the workpiece is inserted and clamped into the collet to establish the primary datum axis. The workpiece may also be bumped up against the face of the collet to locate the secondary datum plane.



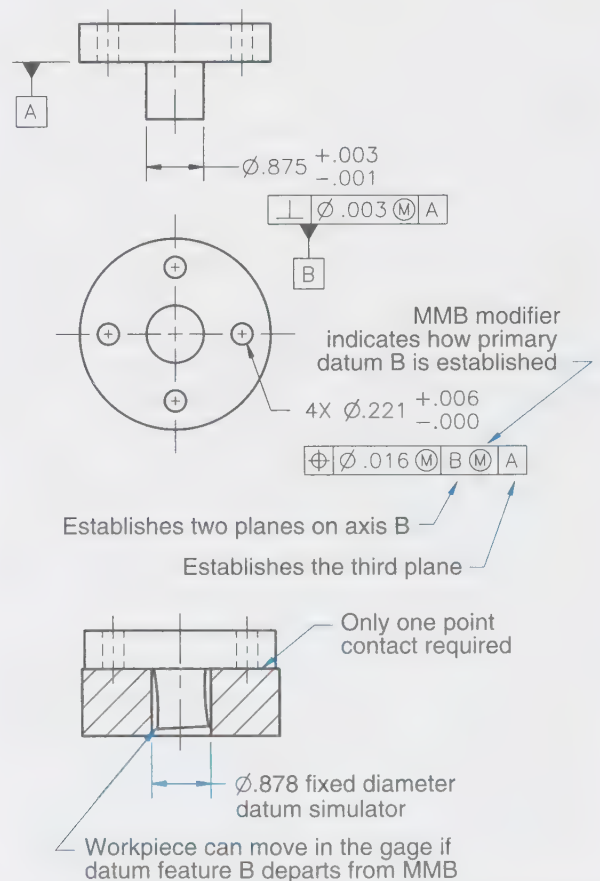
Goodheart-Willcox Publisher

Figure 6-40. A collet may be used to establish a datum axis.

MMB on a Primary Datum Feature Reference

Figure 6-41 shows the same part as the previous figure, except the reference to datum B has an MMB modifier. The MMB modifier applied to a primary datum feature reference has a significant impact on how the datum axis is established. The result is that the datum simulator for datum feature B, on the example part, has a fixed diameter equal to the maximum material size. The MMB and maximum material size are the same for this datum feature on this part.

Three conditions must be met for the MMB of a datum feature to be equal to the maximum material size of that feature. One condition is that the datum must be referenced as a primary datum. The second condition is that the reference must include an MMB modifier. The third condition is that no derived median line (axis) or derived median plane (center plane) form tolerance may be applied to create a virtual condition for the feature.



Goodheart-Willcox Publisher

Figure 6-41. A primary datum feature referenced at MMB requires the datum be simulated by a tool that has a fixed size equal to the maximum material size of the datum feature if only size tolerance is applied to the feature.

In the given figure, datum feature B is referenced as a primary datum and the reference includes an MMB modifier, and no derived median line or derived median plane form tolerance is shown. Datum axis B is therefore located at the center of a simulator or tool that has a diameter equal to the maximum material size of the shaft. The maximum material size of the datum feature is .878" diameter. To locate the axis, the shaft is inserted in a .878" diameter hole. A shaft made .878" diameter will fill the hole, and the axis of the hole and shaft will coincide. Any shaft made at a smaller diameter is free to move inside the hole. The amount of movement depends on the amount of feature size departure from the MMB.

The MMB modifier allows simulation of the datum axis with a fixed diameter gage. This can, especially for high volume production, result in an easier and less expensive inspection process than the axis simulation methods required by a datum feature reference made at RMB.

The three datum feature references in the given position tolerance in **Figure 6-31** define all three planes in the datum reference frame. In **Figure 6-41**, only two datum feature references are needed to establish an adequate datum reference frame. Datum axis B establishes the location of two planes. Datum A locates the third plane.

Depending on the produced diameter of the shaft, datum feature B may or may not establish orientation of the datum reference frame before datum feature A comes into contact with the datum simulator. Consider a shaft made at .878" diameter. If the .878" diameter shaft is made with zero perpendicularity variation, datum surface A will be able to sit flat on its simulator. If the .878" diameter shaft is made with a perpendicularity variation of .003", datum surface A cannot sit flat on its simulator. Both conditions are acceptable based on the tolerance specification showing datum B as the primary datum at MMB.

A reference to a primary datum feature of size at MMB is appropriate for some of the possible applications of the given part. An MMB reference can be appropriate when a clearance fit exists between mating parts, and assembly of the parts is the main concern.

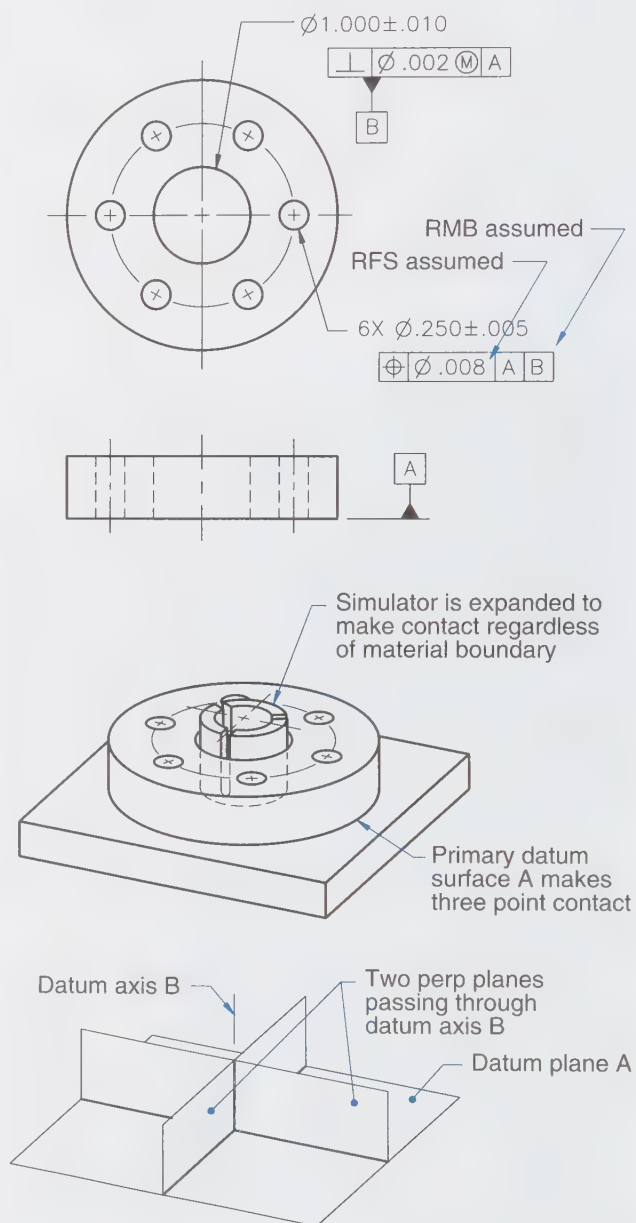
RMB on a Secondary or Tertiary Datum Feature Reference

A reference to a datum feature of size includes a material boundary modifier regardless of whether the referenced datum feature is primary, secondary, or tertiary. It is also applicable when the datum feature is a surface that has a maximum and

minimum material boundary. RMB is assumed for all datum feature references on all tolerances unless the MMB or LMB symbol is shown.

The given part in **Figure 6-42** has a position tolerance with a reference to primary datum A. It does not include a material boundary modifier and is therefore assumed to apply RMB. The reference to the secondary datum feature B also shows no modifier and is assumed to apply RMB.

The first plane in the datum reference frame for the shown position tolerance is established by datum surface A. It is constrained in translation along one axis, and constrained in rotation around



Goodheart-Willcox Publisher

Figure 6-42. A secondary datum feature referenced at RMB requires the datum be simulated by contacting the surface of the datum feature.

two axes. As a result, the rotation of the other two planes in the datum reference frame is constrained, and the planes are required to be perpendicular to datum plane A.

Datum feature B is a hole passing through the part. Datum axis B is perpendicular to datum A and it is located by the *related actual mating envelope* of the hole. In this case, the related actual mating envelope is a cylinder that must be perpendicular to the primary datum and it is expanded until it makes maximum possible contact without pulling the part off the primary datum. Datum axis B is at the axis of the related actual mating envelope. Datum axis B locates the two remaining planes in the datum reference frame (the axis constrains two translational degrees of freedom). Datum axis B must be perpendicular to datum plane A regardless of any variation in the perpendicularity of the hole.

Pro Tip

Mating Envelope Definitions

Actual mating envelope. A theoretically exact geometric counterpart of the feature that is at the maximum size that will fit within the hole or other internal feature or at the minimum size that will encompass a shaft or other external feature.

Related actual mating envelope. An actual mating envelope that is related to one or more datums as required for the geometric tolerances that are applied to the feature.

Unrelated actual mating envelope. An actual mating envelope that is not related to any datums and that encompasses all size and form variations on the feature.

The datum reference frame for the given part may be established through the following datum simulation methods. Primary datum feature A is placed against a flat surface to locate datum plane A. An expanding mandrel extending perpendicular to the flat plate is expanded until it makes contact with the hole. It must expand only until the part cannot slide on the flat plate. This establishes the location of datum axis B. Datum axis B is not required to coincide with the derived median line of the hole because the axis must be perpendicular to the primary plane, and the hole may not be perfectly perpendicular to the primary plane because it has a perpendicularity tolerance of .002" diameter applied to it.

Care must be taken to expand the mandrel just far enough to stop the part from moving. Expanding it too far will align the center of the hole with the mandrel axis and raise the primary datum feature off the datum simulator. If this happens, the datum reference frame is not properly established.

As has been shown above, secondary datum feature references at RMB are simulated differently than primary datum feature references at RMB. As previously explained, a primary datum feature reference at RMB requires that both translation and rotation of the datum relative to the part, or the part relative to the datum, be constrained. A secondary or tertiary datum feature reference at RMB may only constrain the degrees of freedom that are not already constrained by the higher precedence datums. Any datum feature reference made at RMB requires that the feature surface be contacted to establish the datum reference frame.

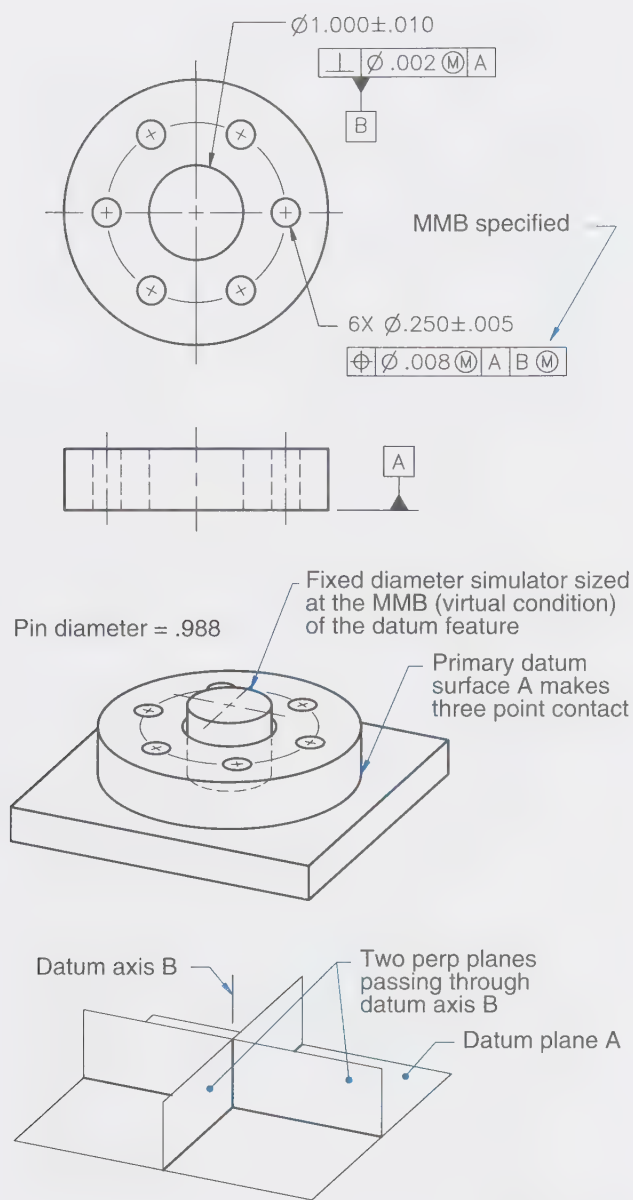
MMB on a Secondary or Tertiary Datum Feature Reference

A reference to a datum feature of size includes either an implied or an explicitly specified material boundary modifier regardless of whether the referenced datum is primary, secondary, or tertiary. When the datum is to apply at MMB, the modifier must be shown. See [Figure 6-43](#).

The given part has a position tolerance with a reference to primary datum A. It does not show a material boundary modifier for the datum feature A reference. The reference to secondary datum feature B includes an MMB modifier.

The first plane in the datum reference frame for the shown position tolerance is established by datum surface A. The primary plane is constrained in translation (located) along one axis, and constrained in rotation around two axes. As a result, the rotation of the other two planes in the datum reference frame is constrained, and the planes are required to be perpendicular to datum plane A. The locations of the second and third planes are established by datum axis B. Datum axis B is perpendicular to datum plane A and located at the center of the simulator for the datum B hole.

Datum plane A is established by a simulator that is a flat plate. Datum axis B is established by a simulator that is a fixed diameter cylinder. The simulator in this example is a fixed-diameter pin that goes through the hole. The pin must be perpendicular to the plate, and it must have a diameter equal to the maximum material boundary of the hole. *A reference to a secondary or tertiary datum at MMB requires that the feature be picked up at its*



Goodheart-Willcox Publisher

Figure 6-43. A secondary or tertiary datum feature reference showing an MMB modifier requires the datum to be simulated by a tool that has a size equal to the applicable maximum material boundary of the datum feature.

MMB, not its MMC size. The maximum material boundary for a hole is equal to the virtual condition resulting from the combined effects of the maximum material size and any applied geometrical tolerance that is related to the higher precedence datum.

The maximum material boundary for secondary datum feature B on the given part is .988" diameter. This is determined by subtracting the .002" perpendicularity tolerance from the .990" maximum material size limit of the hole. The tolerance is subtracted because an out-of-perpendicular

hole would appear to be smaller when fitting over a perpendicular pin.

A condition similar to the previous description for the maximum boundary of a hole also exists for external features such as shafts. The MMB for a shaft is caused by the effect of the maximum material size and any applicable tolerance that tends to make the shaft appear to have an increased diameter. To determine the MMB for a shaft, any applicable tolerance is applied to the MMC size of the shaft. The MMB for a datum feature of size is the same as the virtual condition of a toleranced feature of size as described in detail in following chapters.

It is sometimes necessary to reference one plane and two features of size in order to establish the needed datum reference frame. See [Figure 6-44](#). Four holes are shown with a position tolerance that is related to datum A primary, B secondary at MMB, and C tertiary at MMB. Primary datum feature A is a flat surface. Secondary datum feature B is a hole. Datum feature C is the width of the slot.

Datum simulators are shown in the figure to illustrate how the datum reference frame may be established for this part. Datum A is simulated by a flat surface. Secondary datum axis B is simulated by a fixed-diameter pin. The pin diameter must be equal in size to the MMB of the hole, because the hole is referenced as a secondary datum at MMB. Typically, a secondary datum hole used in this manner will have a size and a perpendicularity tolerance applied. The MMC and the perpendicularity tolerance must both be used to calculate the applicable MMB. The datum B simulator pin must be perpendicular to datum simulator A.

Tertiary datum C is simulated by a flat key. The key must have a width equal to the MMB of the slot. Typically, a tertiary datum slot of this type will include a size tolerance and a position tolerance relative to the primary and secondary datums. The MMC and position tolerances must both be used to calculate the applicable MMB. The datum C simulator must be positioned relative to the datum A and datum B simulators and perpendicular to the datum A simulator.

All three datum feature references are required to completely stabilize the given part. Datum A stops (constrains) translation of the part in one direction and also prevents (constrains) rotation around two coordinate axes. Datum axis B locates two planes that intersect at the axis and prevents (constrains) translation along two of the coordinate axes. These planes are perpendicular to datum A, but are free to rotate on datum axis B.

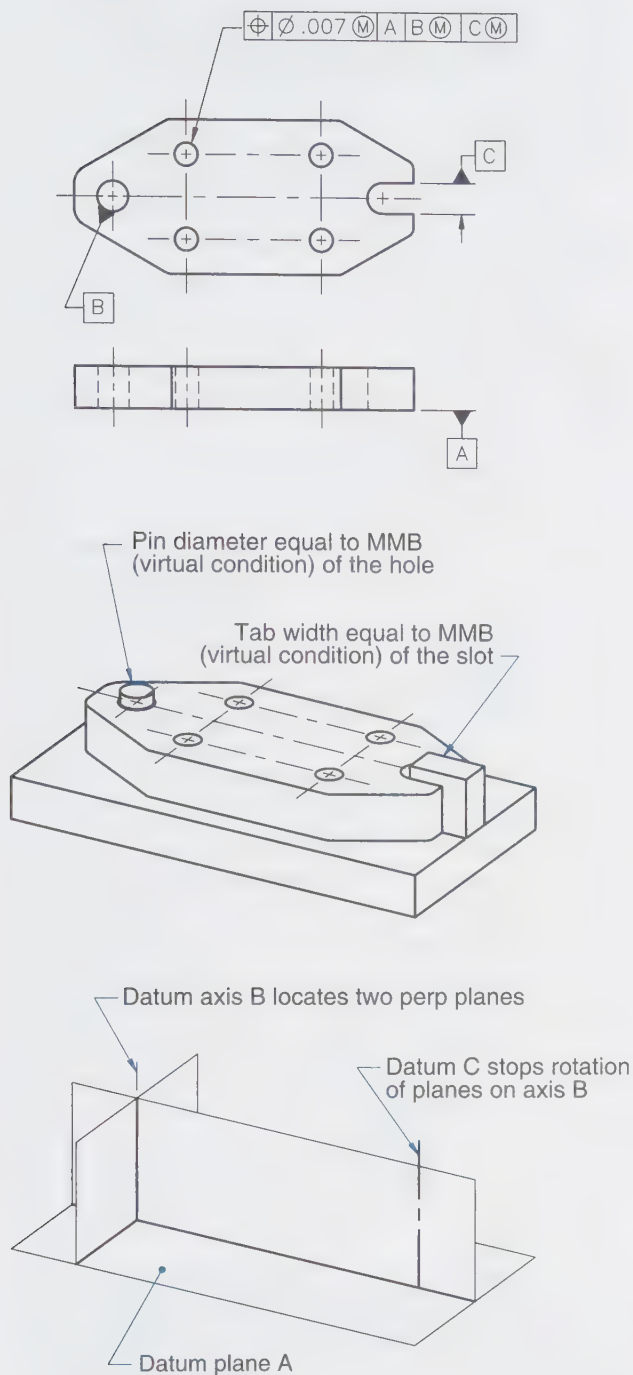


Figure 6-44. A primary datum surface and secondary datum feature of size may be used to establish the datum reference frame with a tertiary datum feature of size used to set the orientation of the datum reference frame.

Datum feature C stops (constrains) the last rotational degree of freedom of the two planes that intersect at datum axis B.

Multiple Feature Control Frames Related to the Same Datum Reference Frame

Multiple groups of features, controlled by multiple single-segment position or profile tolerance

specifications, act as a single pattern when the following conditions are met. The effect of creating a single pattern is often referred to as creating a *simultaneous requirement* or as the *rule of simultaneity*.

1. The feature control frames reference the same datums.
2. The order of precedence for the datums is identical.
3. Each datum feature reference has the same material condition modifier.

Features may be any size or shape and the single pattern requirement applies if the above conditions are met.

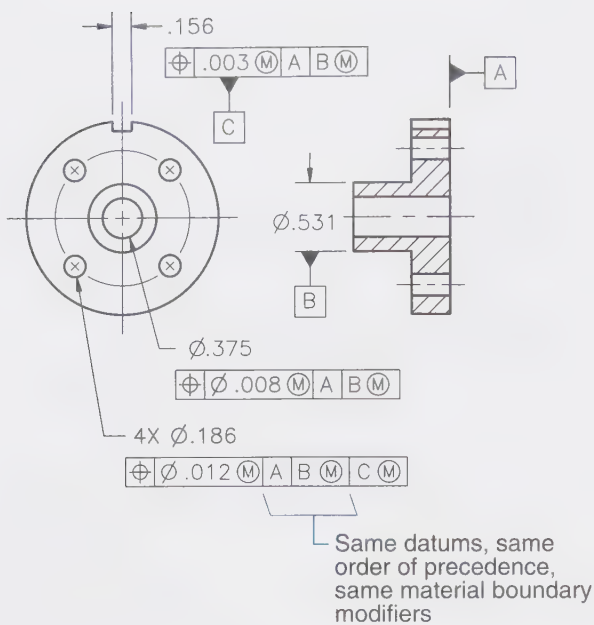
Figure 6-45 shows two drawings that show two means of controlling the positional relationships between the same features. In the first drawing, the slot is controlled relative to datum feature A primary, and datum feature B secondary at MMB. The slot width is identified as datum feature C. The four hole pattern is controlled by a position tolerance referenced to datum feature A primary, datum feature B secondary at MMB, and datum feature C tertiary at MMB. The first two datum feature references in the feature control frames are identical.

However, the feature control frame showing the position tolerance on the four holes includes a reference to tertiary datum feature C at MMB. The intent of this tertiary datum reference is to establish a rotational position of the holes relative to the slot. Establishing a rotational position is typically referred to as *clocking* but technically is a means of constraining one of the rotational degrees of freedom. The reference to tertiary datum feature C is not wrong, but it is not necessary. In fact, referencing datum C at MMB permits some additional variation between the slot and the holes that are position toleranced with datum C referenced tertiary at MMB.

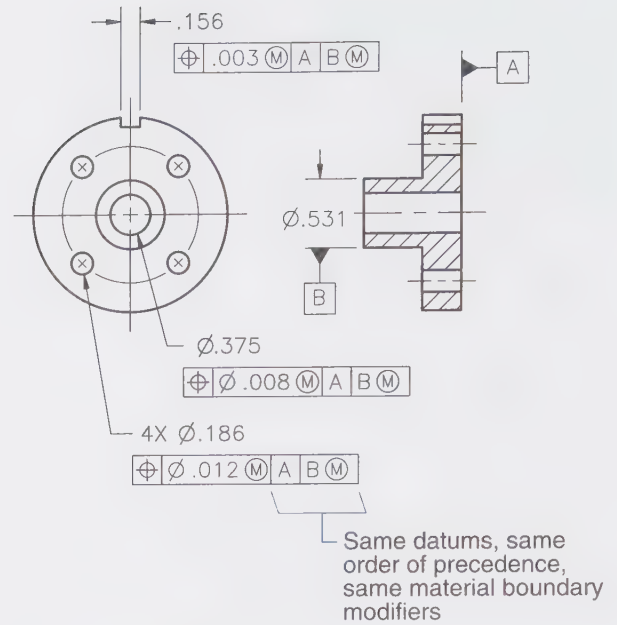
Based on the rule of simultaneity, a rotational relationship between the holes and the slot may be achieved without referencing the slot as datum C. The second drawing shows the same part. Datum C has been omitted from this drawing. Although datum C has been omitted, the relationship between the holes and slot is controlled.

Because the position tolerances applied to the slot and to the four holes have the same datum feature references, in the same order of precedence, and with the same material condition modifiers, the slot and the holes act as a single pattern. Because they act as a single pattern, the rotational position of the holes relative to the slot is controlled.

The principles of groups acting as a single pattern are covered in more detail in the chapters



Datum C is identified and used to clock the hole locations



Datum C is omitted and the single pattern requirement establishes clocking locations

Goodheart-Willcox Publisher

Figure 6-45. Groups of features toleranced relative to the same datum reference frame act as a single pattern.

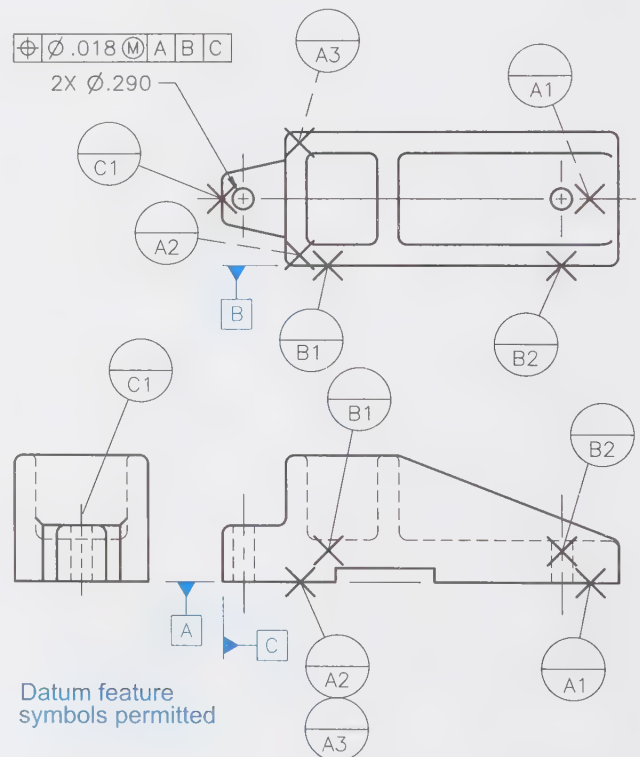
on position tolerances. The same principle applies to profile tolerances, and to a combination of position and profile tolerances.

Datum Targets Based on Order of Precedence

The datum precedence expressed in a feature control frame must be considered when defining datum targets on a drawing. See **Figure 6-46**. A two hole pattern on the given part has a position tolerance that references primary datum feature A, secondary datum feature B, and tertiary datum feature C.

Datum targets are used to identify datum features on the given part because the cast surfaces are considered too uneven for use as datum surfaces. Datum feature symbols may be used in addition to the datum target symbols, but they are not required.

The feature control frame on the given drawing includes a reference to primary datum feature A. Because target points are being used, a minimum of three targets is required to establish primary datum A. For a small rigid part, no more than three targets should be used unless notes are used to explain that only three of the targets must make contact.



Goodheart-Willcox Publisher

Figure 6-46. The number of targets used for each datum feature depends on the order of precedence for the datum.

The given feature control frame includes references to secondary datum feature B and tertiary datum feature C. Secondary datum feature B is identified by two datum target points. Tertiary datum feature C is identified by one datum target line. The target line could be replaced by a target point if desired.

Regardless of whether target points, lines, or areas are used, the datum precedence must be considered prior to deciding the number of targets to show on a feature.

In some applications, not all surfaces are machined when using cast, forged, or molded parts. There are also applications where all surfaces are machined. See **Figure 6-47**. When possible, it is a good practice to locate machined datums from cast datums. If no material is machined from the cast datum features, it is possible to confirm that machined surfaces are properly located relative to the cast datums. If material is removed from the cast datums, it is not possible to measure from them on the final part to determine how well the machined surfaces are located relative to the cast datums.

It is sometimes necessary to use a feature as a primary datum in one tolerance specification, and a secondary datum in another. If datum targets are specified on the surface, the dual use of one datum feature requires some special attention. See **Figure 6-48**. One way of using a single feature for two levels of datum precedence is to identify the surface with two datum letters. One of the datum letters is to identify the primary reference and the second datum letter is for the secondary reference. More than one datum letter should only be applied to a single feature when using datum targets.

Figure 6-48 shows two feature control frames. In the first, datum feature A is referenced

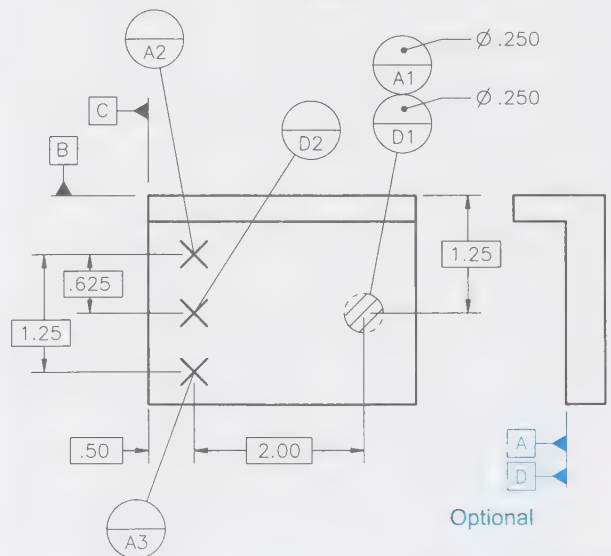
as a primary datum and is identified with datum target points. Because datum feature A is primary, at least three datum target points must be defined. The second feature control frame requires the same surface be used as a secondary datum. To accomplish this, the surface is given a second identification, datum feature D. Datum feature D is a secondary datum and is therefore defined using two target points, one of which shares a common location with target point A1. It is not required that any of the points share a common location, but one or both of them may.

Although the same datum feature may be referenced at two levels of datum precedence in different feature control frames, the same datum letter is not normally used to indicate both the primary reference in one feature control frame and secondary reference in another feature control frame when using datum targets. The reason is simple. If three points are used to establish primary datum A, a reference to the same datum feature A as a secondary reference would not indicate which two of the three points create the secondary datum. However, if any two of the three points are allowable for a secondary datum reference, then a reference to datum feature A would be allowed, although it could potentially result in some confusion. A note could be used to explain the requirements.

There are situations where only the primary datum is needed, such as for a perpendicularity tolerance, yet the target locations for the primary

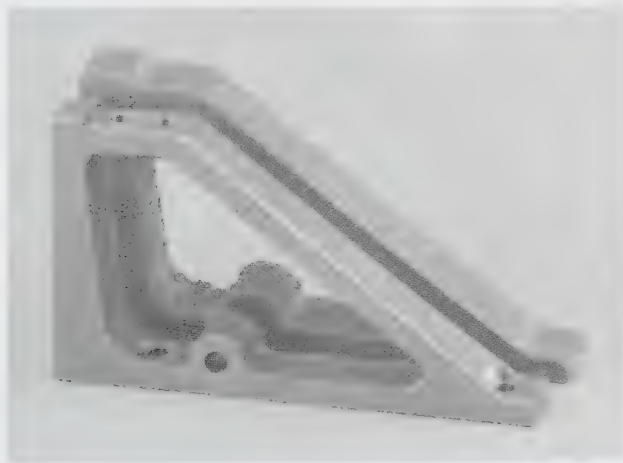


Datums A and D are established by the same feature, but are targeted according to datum precedence



Optional

Figure 6-48. The same surface may be identified with more than one datum letter when using datum targets.



Goodheart-Willcox Publisher

Figure 6-47. Machined datum features are often located relative to cast features using datum targets.

datum feature cannot be established without the use of the secondary and possibly the tertiary datum features. Placing a perpendicularity tolerance on one of the surfaces and referencing only datum feature A would require the datum to be established from the three datum targets A1, A2, and A3. However, the datum feature A targets may only be established relative to datum features B and C. This creates a problem because datum features B and C are not referenced. The solution is not to add the datum references in the perpendicularity feature control frame. That would change the tolerance requirement. The solution to this problem is to add a drawing note similar to the following:

DATUM A IS ESTABLISHED USING DATUM FEATURE B SECONDARY AND DATUM FEATURE C TERTIARY.

Special Applications

Datum surfaces, datum features of size, and datum feature references (as explained in the previous sections of this chapter) meet the needs of many design requirements. However, there are times when datums and tolerances must be applied to meet complex design requirements. Some special applications of datums and datum feature references are described in the remaining portion of this chapter.

Multiple (Compound) Datum Features

Rotating shafts are commonly supported by two or more bearings. Multiple bearing surfaces are functional in those situations and should be used to establish the axis of rotation for the shaft. Specifying multiple (compound) datum features permits the design tolerances to reflect the functional needs of a rotating shaft. The current standard refers to the use of more than one datum feature to create a single datum as *multiple datum features*. To avoid confusion between this particular use of the term *multiple* and any other arrangement utilizing more than one datum feature, this text will use the term *compound datum features*. **Compound datum features** are two or more features used at the same time to establish one datum.

Figure 6-49 shows how compound datum features may be used. Bearing diameters on each end of the shaft are identified as datum features. Each has a datum feature symbol attached. One is identified as datum feature A, and the other is identified as datum feature B.

Three runout tolerance specifications reference datum features A and B in a manner that

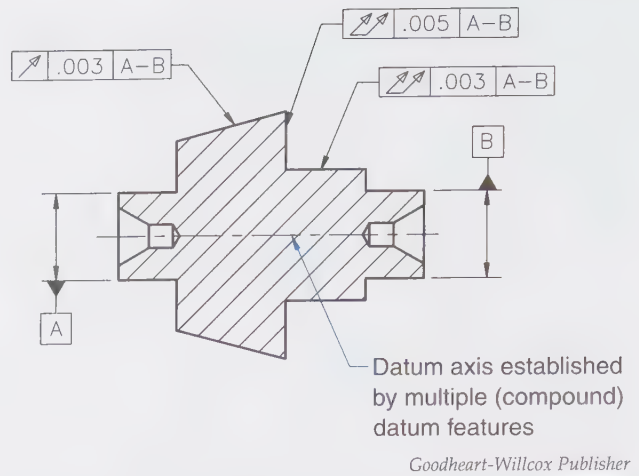


Figure 6-49. A single datum axis may be established by multiple (compound) datum feature reference.

establishes a single datum axis from the two datum features. This is done by showing a compound datum feature reference. A **compound datum feature reference** shows two or more datum letters separated by a dash. The datum letters and the dash(es) are contained in a single cell within the feature control frame. Neither datum feature has precedence over the other.

The datum axis for this part may be simulated by placing each end in a chuck. The axis between the two chucks acts as the datum axis. Another means of establishing the datum axis is to place the two datum features in V-blocks. See Figure 6-50. The V-block method is relatively accurate if the datum features are round. If the datum features are out-of-round, then the V-blocks will not accurately locate the axis.

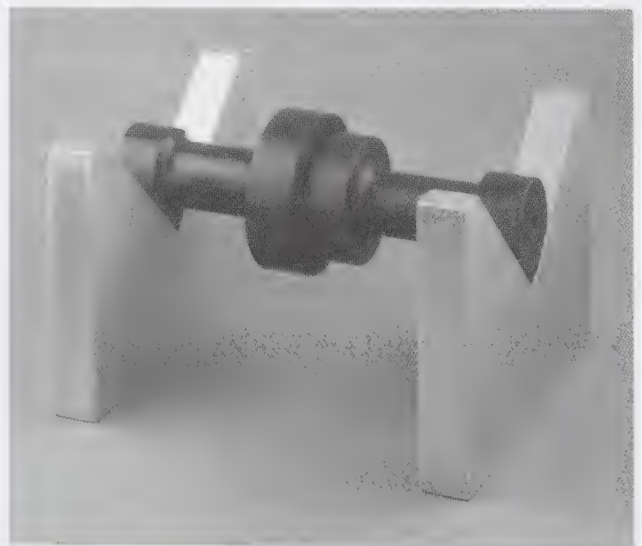


Figure 6-50. V-blocks are sometimes used to establish a datum axis from multiple (compound) datum features.

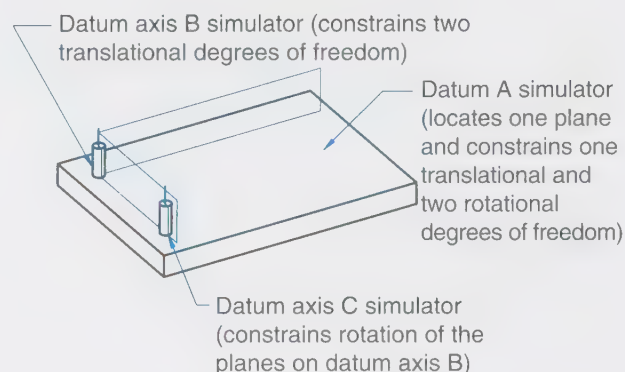
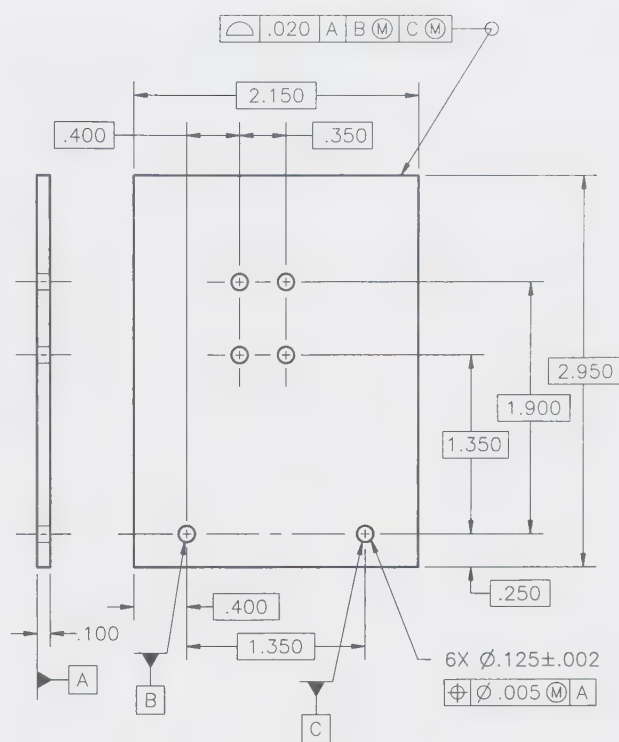
Compound datum features may also be used to establish a datum plane. See **Figure 6-51**. The given part has two slots, one cut in each end of the part. Each slot is identified as a datum feature. The profile tolerance on the curved surfaces references compound datum feature B-C as a secondary datum. The two slots are picked up simultaneously to establish the center plane through them.

Compound datum features may be made of two or more flat surfaces. Each surface is identified as a datum feature and referenced by showing all of the datum feature letters separated by dashes. If the surfaces lie in the same plane, there is more than one way to establish that a single datum is to be established from them. The compound datum feature method may be used, it is possible to use the continuous feature symbol to link them together, and a profile tolerance may be applied to the surfaces and a datum feature symbol applied to the profile tolerance.

A Surface and Two Holes

Flat panels and many other part types often include two holes that locate the part in an assembly, and it is common to use two holes to locate a part during fabrication. See **Figure 6-52**. These holes may be identified as datum features, and tolerances referenced to them.

The given part has six holes. They are all tolerated relative to datum A. The holes act as a single pattern. Two of the holes are identified as datum features B and C. The perimeter of the part has a profile tolerance that is referenced to datums A primary, B secondary at MMB, and C tertiary at MMB. This tolerance completely controls the location, orientation, and size of the perimeter with respect to the specified datums. In effect, the outline of the part is controlled to the positions of the holes.



Goodheart-Willcox Publisher

Figure 6-52. A surface and two holes may be used to create and orient a datum reference frame.

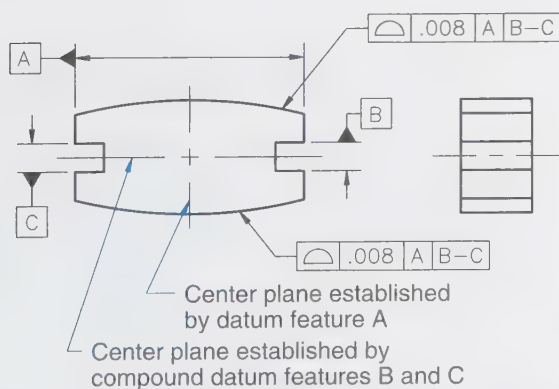


Figure 6-51. Rectangular features of size may be used as compound datum features to establish a datum plane.

Goodheart-Willcox Publisher

The primary and secondary datums in the given example locate all planes in the datum reference frame, but rotation of the second and third planes is not constrained until the tertiary datum is defined. This was previously explained with respect to **Figure 6-44**.

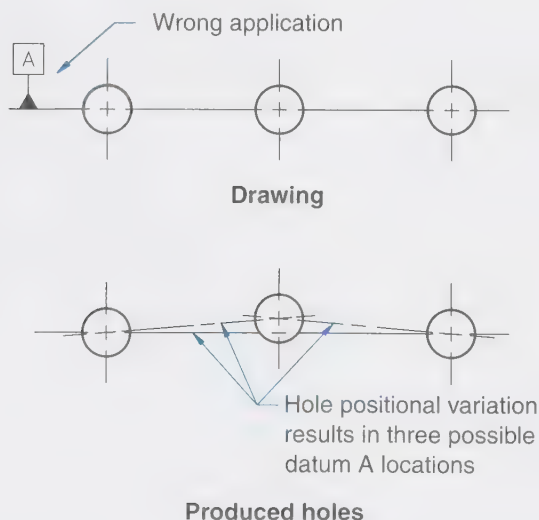
Figure 6-52 shows that a datum reference frame may be completely defined by one surface and two holes. The datum features used are physical features on the part. It is important that the datum reference frame is created by referencing features on the part. Datum feature symbols are not placed on the drawing centerlines between or through the holes.

One significant reason for not placing datum feature symbols on centerlines is that the standard specifically says not to do it. There is no defined meaning for a datum feature symbol placed on a centerline. The absence of a defined meaning should be adequate for not placing datum feature symbols on centerlines.

Another reason for prohibiting the placement of datum feature symbols on centerlines is the confusion it can cause. Consider the placement of a datum feature symbol on a centerline that passes through three holes. See **Figure 6-53**. The figure shows a centerline through three holes, and it is identified as datum A. Because the datum feature symbol is on the centerline, it is not known which holes to use for the location of datum plane A. Should all three holes be used, or should only two be selected? If two are used, which two?

It is important to realize that three holes may not be produced perfectly on one line. When the produced holes are not in a straight line, two of them must be selected to establish the datum. The given figure shows the effect of arbitrarily selecting two holes to establish datum A. (Selection must be arbitrary because the drawing does not indicate which holes to use.) Selecting two holes at a time results in three possible datums. Which pair of holes should be used? This question does not have an answer because the datum feature symbol is not properly applied. One correct method for identifying datums that pass through multiple holes is shown in **Figure 6-52**.

Another common and confusing mistake is the application of datum feature symbols on two



Goodheart-Willcox Publisher

Figure 6-53. It is incorrect to show a datum feature symbol on a centerline that passes through a group of holes.

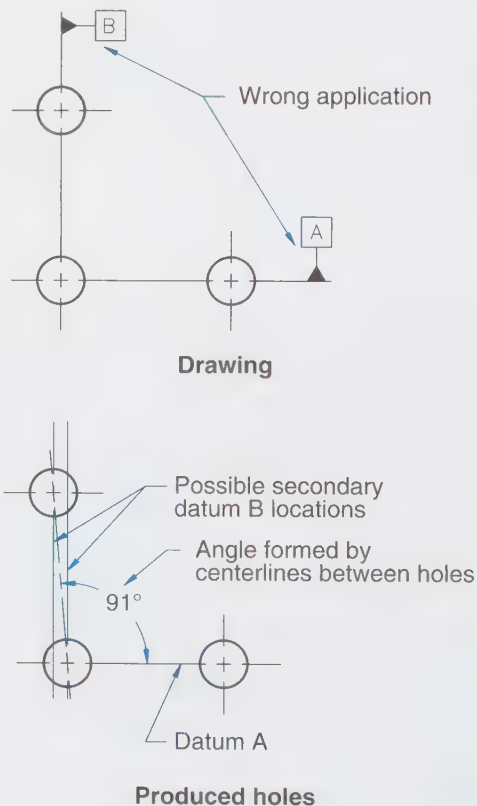
perpendicular centerlines. See **Figure 6-54**. In the given example, two perpendicular centerlines pass through three holes. The centerlines on the drawing intersect at one of the holes, and each centerline passes through one of the other two holes.

Identification of two perpendicular lines that are supposed to run through three holes causes problems. Holes are seldom, if ever, produced at the exact dimensioned locations relative to one another. It is likely that some variation from the basic dimensions will exist in the location of the produced holes.

The given figure shows three produced holes. They are within the allowable position tolerance relative to one another, but lines drawn between the holes do not form a perfect 90° angle. Because a coordinate system has 90° between axes, all three holes cannot be used to establish two planes of the coordinate system.

Two holes are all that are needed to establish a coordinate system and constrain the translation and rotation of that system. This was shown in **Figure 6-52**.

Because one hole establishes the origin (stops translation) and another hole stops rotation of the coordinate system, attempting to use a third hole



Goodheart-Willcox Publisher

Figure 6-54. Drawing requirements are not clear when datum feature symbols are placed on two perpendicular centerlines.

only raises questions. What should be done if the third hole does not rest on the established coordinate system? Should the coordinate system be distorted to some angle other than 90° just to use the specified hole? Of course not; therefore, it is incorrect to use the third hole for this purpose. If three holes are used and the coordinate system must maintain the 90° angle, there will be a problem in deciding what holes to use. As shown in the figure, it would not be clear whether the origin should be established by one of the holes, or by perpendicular planes passing through the holes.

Avoid using three holes to define a coordinate system. Using three holes raises questions for which there are no defined answers in the ASME dimensioning and tolerancing standard. The problems and ambiguity of using three holes to establish a coordinate system can be avoided by correctly using the methods illustrated in [Figure 6-52](#).

Stepped Datum Surfaces

Parts are often designed that must mount on stepped surfaces. Because datum features are normally selected on the basis of how a part is assembled, it is desirable to have a means for using offset or stepped datum surfaces. See the example in [Figure 6-55](#).

Stepped datum surfaces are the condition of two or more non-coplanar surfaces used to establish a single datum. When stepped surfaces are used to define a datum plane, the step distance is

given with a basic dimension. The basic dimension defines the step distance for the datum simulator (the tooling or simulation method) that establishes the datum plane. In the given example, datum target points are used to define the stepped datum requirements and the distance between targets is specified with basic dimensions.

The distance between the surfaces on which the stepped datum targets are located may be indicated with the same basic dimension that defines the distance between the stepped datum targets. If the basic dimension defines both the target step distance and the distance between surfaces, a profile tolerance is typically required to define the surface location tolerance. The target location tolerance is not defined on the drawing. Standard toolmaking tolerances are understood to apply for the location of targets dimensioned with basic numbers.

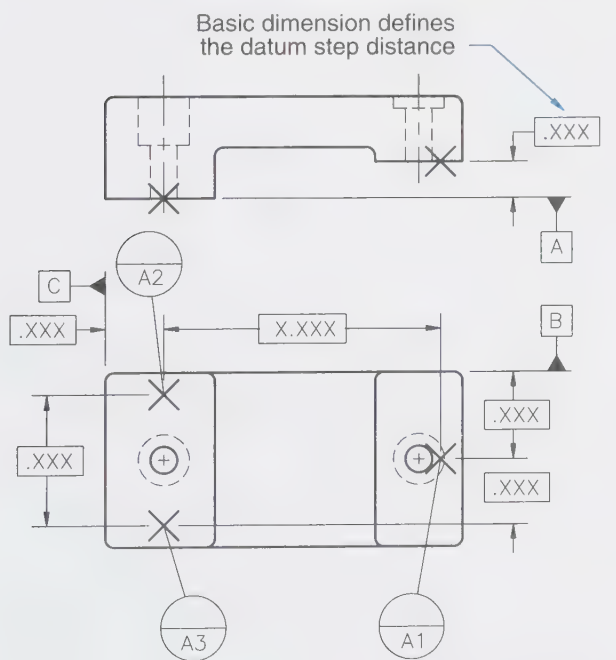
Equalizing Targets to Locate Datum Planes

Castings and forgings are often designed such that datum planes need to be located through the use of equalizing datum targets. Equalizing targets are also utilized for parts made using other processes. *Equalizing datum targets* cause the workpiece to be centered when the targets are contacted. See [Figure 6-56](#).

Four datum targets are used to locate datum B on the given part. Targets B1 and B2 are horizontal target lines at a 90° angle to each other. Targets B1 and B2 are known to be target lines because of the horizontal lines shown in the front view. Removing the horizontal lines from the front view would change the target requirements to planes at a 90° angle.

Datum plane B bisects the 90° angle formed by the target lines B1 and B2 and passes midway between targets B3 and B4. Targets B3 and B4 are vertical target lines and are identified with the movable datum target symbol. A **movable datum target symbol** is a means of indicating the datum target simulator is movable. Targets B3 and B4 move simultaneously to make contact with the part. Movable datum targets default to move normal to the workpiece surface if the direction is not indicated. However, the targets on the shown part are located with basic dimensions in only one direction and therefore may only move in one direction.

If an additional dimension was shown or the location defined without dimensions in a solid model, there would be a fixed location in space for the target. When a specific target location is



Goodheart-Willcox Publisher

Figure 6-55. Stepped datums permit offset features to be used to locate a single datum plane.

define the features that establish the datum plane. If it is desired for clarity, a notation may be applied to indicate the datum plane location that is established by the targets.

Targets on Diameters

It is not always possible to use the entire surface of a cylindrical part as a datum feature of size. Datum targets may be identified on cylindrical features. See [Figure 6-57](#). The given part has six datum target lines, and they are used to define datum axis A. There are three target lines equally spaced on each end of the part. The targets are identified as A1 through A6. They are shown in both views to provide a clear definition of the target definition. The datum target simulators are required to move normal to the surface until contact is made with the datum feature when the datum feature is referenced RMB. The movable datum target symbol may be used but is not required.

Targets placed on a cylinder must take into consideration how the targets can be contacted. The part in this figure may be clamped on each end with a three-jaw chuck. Each jaw will make contact with a line along the targeted surface.

When the datum axis is defined by two features having different diameters, then the targets on each feature shall be given separate datum feature letters. See [Figure 6-58](#). It is possible to use datum target points, lines, or areas to establish a datum axis. They may also be used in combination.

The given part has datum targets B1, B2, and B3 on one feature. On the other end of the part, datum target area A1 is shown. The datum axis

for this part may be referenced by placing the multiple (compound) datum feature reference A-B in a tolerance specification.

An alternate datum targeting method using a target line is shown in [Figure 6-58](#). The target line is shown across the cylinder, but the target actually is a circular element on the surface of the part.

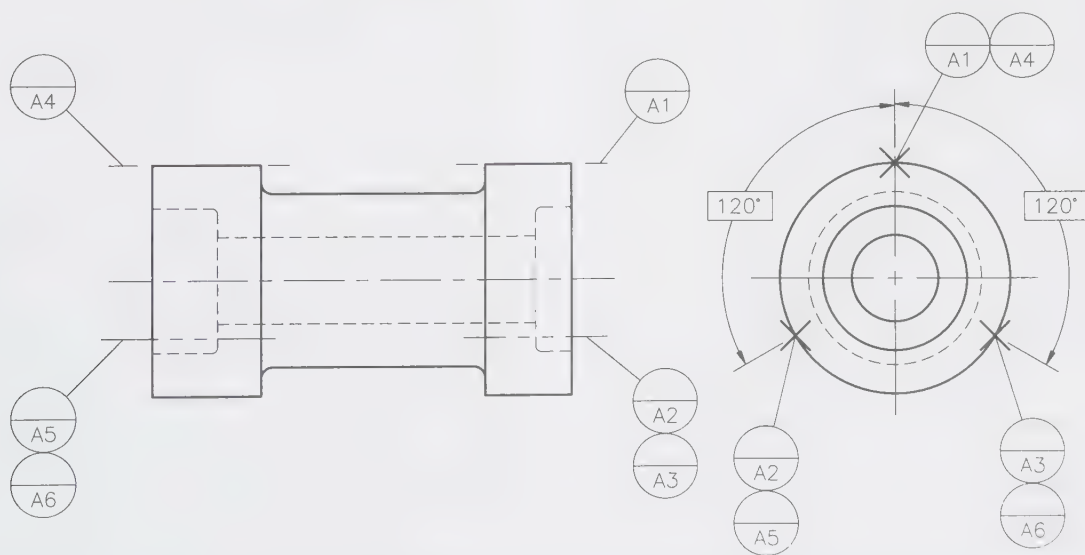
The targets shown in [Figures 6-57](#) and [6-58](#) are on external features. The same techniques may be applied to internal features.

Combined Target Areas and Points

Datums are defined by the means that best reflects the functional requirements of the part and also facilitates manufacturing. This often means using datum targets, and sometimes results in combined target types.

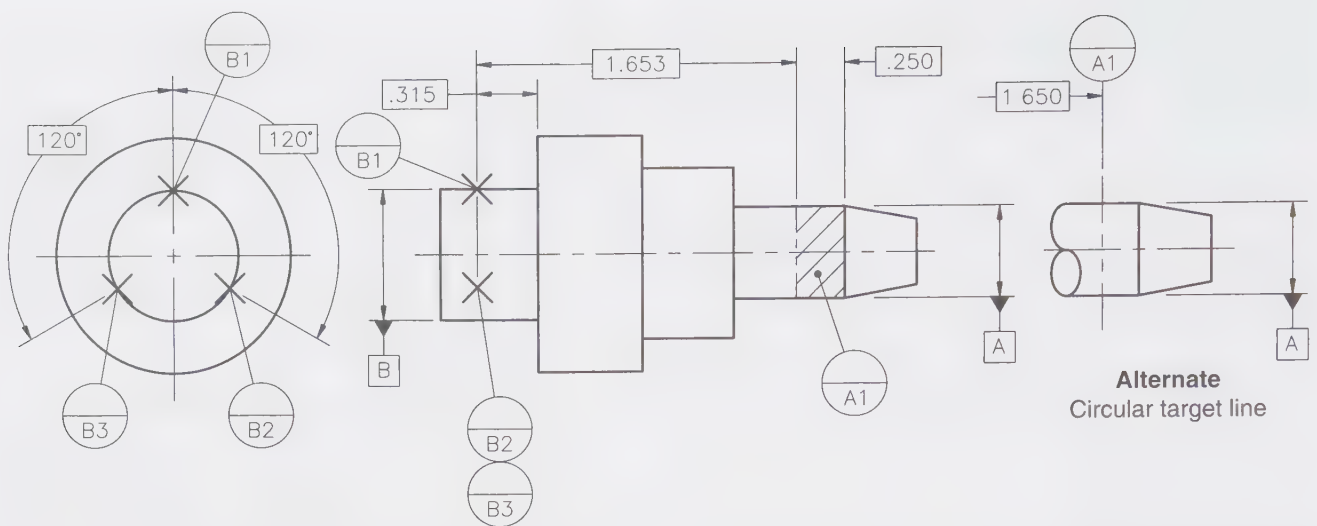
Target areas may be combined with points as shown in [Figure 6-59](#). Large target areas may result in multiple point contact at random locations across the area, similar to when an entire surface is identified as a datum feature. Datum target area A2 on the given part is relatively large. It spans the full depth of the part. Combining target area A2 with target point A1 ensures at least three points of contact, should datum A be referenced as primary.

Target lines may be combined with areas, and points may be combined with lines. Whatever combination best represents the functional requirements is appropriate. It is necessary for any selected combination to provide the proper datum location based on the order of precedence for the datum.



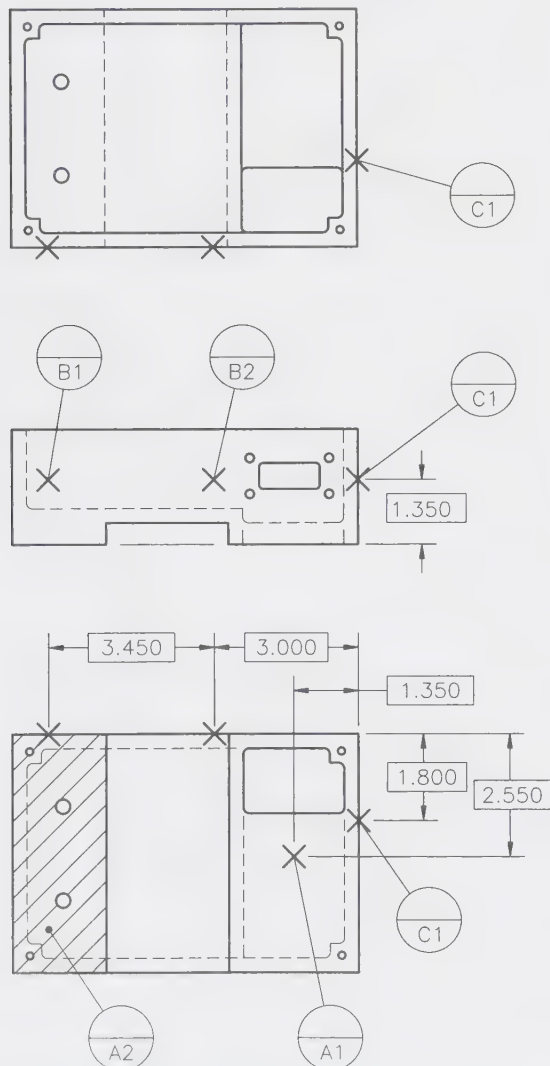
Goodheart-Willcox Publisher

Figure 6-57. Datum targets may be used to establish a datum axis.



Goodheart-Willcox Publisher

Figure 6-58. Datum target areas may be used to define the area of a cylinder that must be contacted to locate a datum axis.



Goodheart-Willcox Publisher

Figure 6-59. Datum target types may be mixed to locate a single datum plane.

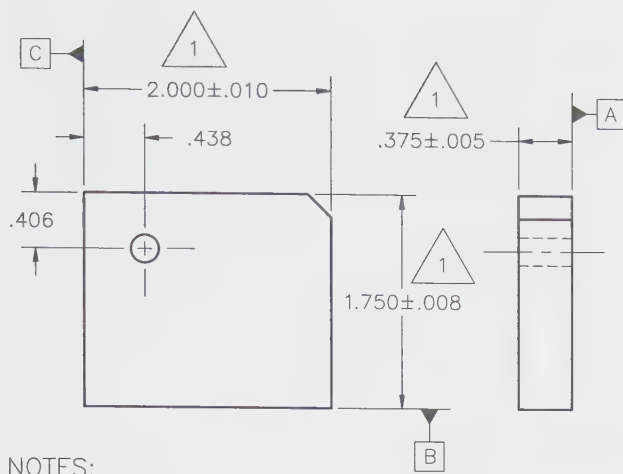
Dimensions Related to a Datum Reference Frame

As explained in Chapter 4 of this text, Rule #1 of ASME Y14.5 requires perfect form of a feature when it is at maximum material condition. The requirement for an individual feature to have perfect form does not require the feature to be in any specific relationship to other features. Relationships between features are normally specified in tolerance specifications such as orientation, position, and runout.

It is also possible to control feature size and orientation relative to a datum reference frame. See [Figure 6-60](#). This is done by including a note on the drawing. The note may be general and apply to all dimensions, or it may be a local note that is applied to specific dimensions. If a general note is used, make sure that dimensions extend from the datum features. The note must specify the order of precedence for datums to which dimensions are referenced.

A local note is applied to the given drawing. It states that NOTED DIMENSIONS ARE RELATED TO DATUM A PRIMARY, DATUM B SECONDARY, AND DATUM C TERTIARY. The note is placed in a notes list, and the note number is placed adjacent to the applicable dimensions.

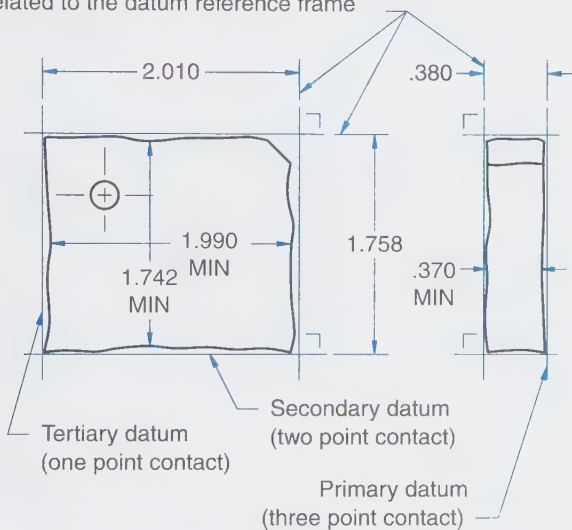
The effect of dimensions related to a datum reference frame is the creation of an envelope. An *envelope* is a geometric boundary that contains an external feature or is within an internal feature. Normally, an envelope, when created by tolerances, establishes one boundary that may not be violated. Envelope measurements for the given figure are made relative to the datum reference



NOTES:

1 NOTED DIMENSIONS ARE RELATED TO DATUM A PRIMARY, DATUM B SECONDARY, AND DATUM C TERTIARY.

Perfect form envelope at MMC related to the datum reference frame



Goodheart-Willcox Publisher

Figure 6-60. Dimensions related by note to the datum reference frame establish an envelope at the MMC size of the dimensioned feature.

frame. Envelope size is based on the maximum material condition for the dimensioned feature. The envelope has perfect orientation with respect to the noted datum reference frame.

The requirement for perfect orientation to the datum reference frame could potentially increase product cost above the requirements of Rule #1, which does not by itself force perfect orientation relative to the datum reference frame. Relating dimensions to a datum reference frame should only be used when design requirements dictate that it is necessary.

The impact on the least material condition requirements when dimensions are referenced to a datum reference frame is somewhat controversial.

A generally accepted approach is to verify least material limits by direct two-point measurement between surfaces. This approach ensures that the part features do not violate the minimum size limit allowed by the dimension tolerance.

Advanced Datum Concepts

Advanced datum concepts allow a new level of clarity in specifying complex part requirements. Also, defining how datums are established from the datum features that are identified and referenced is improved.

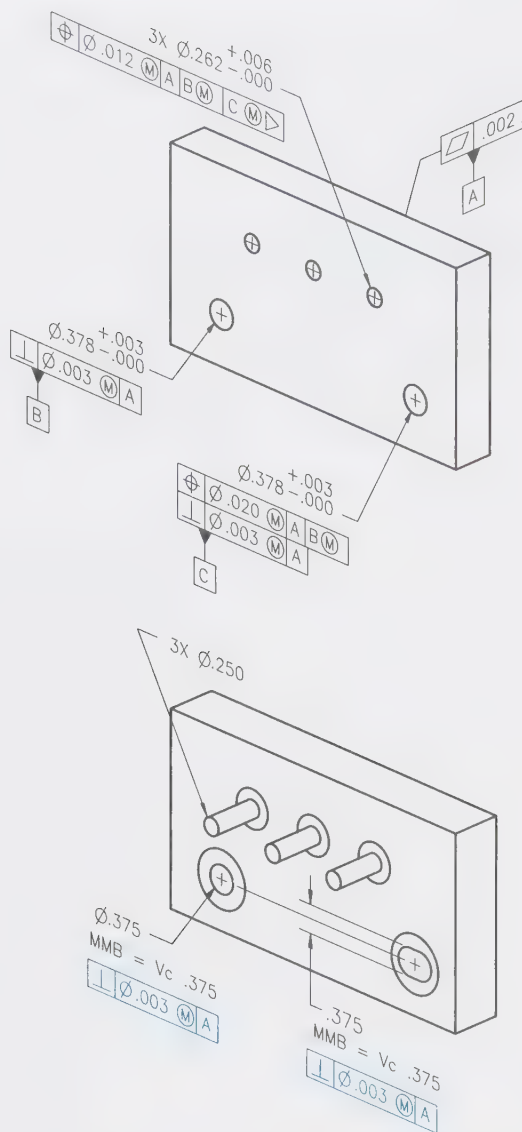
Datum Feature Translation

It is possible to indicate in the datum feature references that the datum simulator may translate. See [Figure 6-61](#). The drawing of the plate shows one hole as secondary datum feature B and another hole as tertiary datum feature C. The hole identified as datum feature B has an orientation tolerance that is related to datum feature A. The hole identified as datum feature C has two geometric tolerances applied to it. It has an orientation tolerance relative to datum A and a position tolerance that references datum feature A primary and datum feature B secondary at MMB.

Three holes in the plate have a position tolerance that references datum feature A primary, datum feature B secondary at MMB, and datum feature C at MMB with the translation modifier shown. Primary datum feature A is simulated with a flat plate. Secondary datum feature B is simulated at the applicable MMB size of .375" diameter, which is the result of the maximum material size and the orientation tolerance that is applied to the datum feature B hole. The tool drawing shows a round bushing is used to locate a .375" diameter tooling pin that would be inserted through the datum feature to establish the orientation and location of datum axis B.

Simulation of the tertiary datum feature must be done using the applicable MMB. Because the translation symbol is shown on the tertiary datum feature reference, the simulator does not need to accommodate the position tolerance because the simulator is permitted to translate. The applicable MMB for datum feature C is determined using the maximum material size and the perpendicularity tolerance. The maximum material size of .378" diameter and the perpendicularity tolerance of .003" diameter combine to establish an MMB size of .375" diameter for the simulator.

In the given figure, a slotted bushing is used to locate the .375" diameter tooling pin that would



Goodheart-Willcox Publisher

Figure 6-61. The translation symbol applied to a datum feature reference permits the simulator to move in a manner that is appropriate in relation to higher precedence datums.

be inserted through the datum feature to establish the tertiary datum axis. It should be noted that the slotted bushing cannot verify the perpendicularity or position tolerances applied to datum feature C. The slotted bushing only serves to correctly establish datum C as it is referenced in the position tolerance for the three holes.

Customized Datum Feature References

There are six degrees of freedom including x, y, z translations along the coordinate axes and u, v, w rotations around the coordinate axes. These six degrees of freedom are constrained to the extent required by the datum feature references in a feature control frame. Each datum feature reference

constrains all possible degrees of freedom that are not already constrained by higher precedence datums and that can be constrained by its geometry.

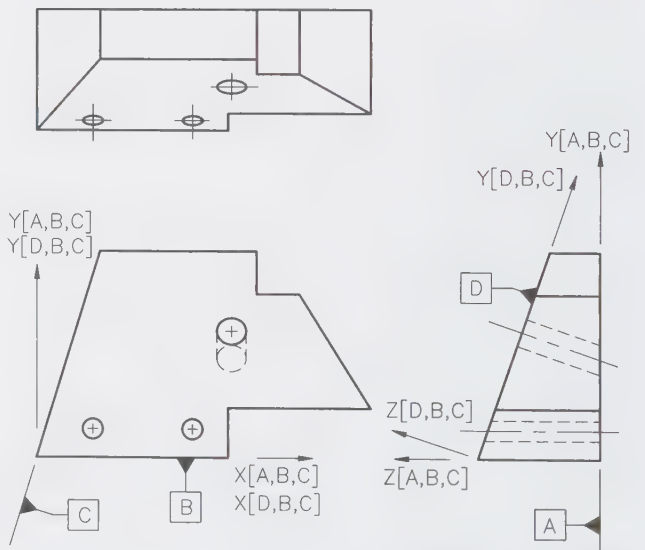
Wherever it is necessary to take exception to the default constraints of the datum references, the desired degrees of freedom are noted following each of the datum feature references. The notations are made using lowercase letters. For these notations to have meaning, the coordinate axes are identified by labeling them on the drawing or in the CAD model. Uppercase letters are used for axis labels.

Noting Translational Degrees of Freedom

The coordinate axes X, Y, and Z are labeled, when needed, using uppercase letters. The translational and rotational degrees of freedom are not typically labeled because the translational degrees of freedom are understood to be parallel to the axes with the rotational degrees of freedom around those axes. See [Figure 6-62](#). When there are two or more coordinate axis systems created by multiple datum reference frames, then the applicable datum reference frame is noted adjacent to each coordinate axis direction label. The applicable datum reference frame is placed within brackets.

Specifying Applicable Degrees of Freedom in Datum References

It is not always desirable for a datum feature to constrain all the possible degrees of freedom. Should it be desired to limit the degrees of freedom constrained by a datum feature reference, then the applicable degrees of freedom need to be



Goodheart-Willcox Publisher

Figure 6-62. The applicable datum references are noted when labeling the axes of more than one datum reference frame.

shown on all the datum references in the feature control frame. The constrained degrees of freedom are placed within brackets and the degrees of freedom are noted using lowercase letters. See [Figure 6-63](#). The given feature control frame includes a reference to primary datum feature A with it constraining the z, u, and v translation and rotations. The reference to secondary datum feature B requires that it only constrain the w rotational degree of freedom. If the notation had not been shown, the secondary datum feature would have also constrained y translation. Because the notation of y is not included, that degree of freedom is not constrained by the secondary datum feature. The tertiary datum reference includes notations that require it to constrain both the x and y translations. Care must be taken to make sure the notations that follow a datum feature reference are degrees of freedom that can be constrained by the geometry of the datum feature.

RMB and MMB on Surfaces

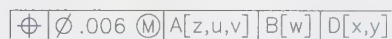
When a surface is controlled with a profile tolerance, the profile tolerance creates boundaries for that surface. The boundary farthest outside the material creates the maximum material boundary (MMB) and the boundary the farthest inside the material is the least material boundary (LMB). Any feature that has material condition boundaries may be referenced as a datum feature and it is assumed to apply RMB if not indicated otherwise. The MMB or LMB modifier may be specified where needed. This concept is the same regardless of whether the boundaries exist for a feature of size or a surface.

Pro Tip

Datum Target Symbol Leader

When a profile tolerance is applied to a curved surface, referencing the curved surface as a primary datum RMB creates a requirement that is difficult to simulate. The datum simulator is supposed to progress through the tolerance zone until maximum possible contact is achieved. To avoid this difficult simulation requirement, the datum reference may be made at basic. Another possibility is to apply datum targets on the surface because the datum targets are to be simulated at their basic locations.

Previous sections in this chapter introduced the application of RMB, MMB, and LMB for features of size and RMB and MMB application on



Goodheart-Willcox Publisher

Figure 6-63. The required degrees of freedom are noted on each datum feature reference when customizing a datum reference frame.

surfaces. Because surfaces do not have size limits, the application of material boundary modifiers on surfaces may cause some confusion and will be further explained in the following paragraphs.

The effects of material boundary modifiers on datum features of size will be further explained and then the same concepts will be explained for datum feature surfaces.

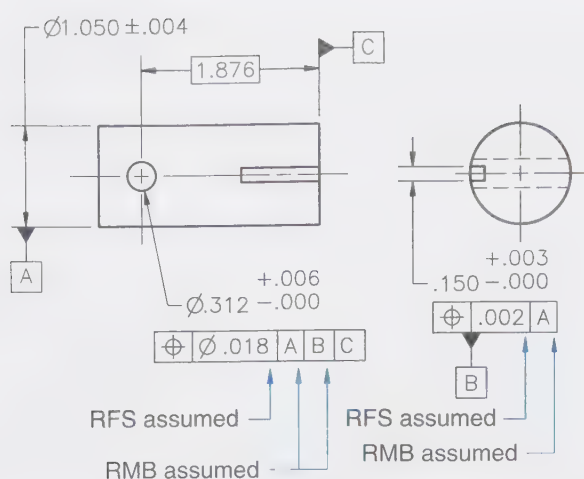
RMB, MMB, and LMB Application to Datum Features of Size

[Figure 6-64](#) shows a shaft with a keyseat and hole. The shaft diameter, keyseat, and one end surface are identified as datum features. The keyseat and hole have position tolerances applied. The material boundary modifiers applied to the datum feature references have a significant effect on the tolerance requirements, and both the RMB and MMB effects are shown.

The shaft diameter is a feature of size identified as datum feature A. It has only a size tolerance applied; therefore, it has perfect form at its maximum size because of Rule #1. The keyseat is also a feature of size and has a position tolerance that references datum feature A at RMB. The keyseat has both a size tolerance and position tolerance that create a maximum material boundary. The hole has a size tolerance and position tolerance applied to it.

There are many possible datum reference frames that could be created by using different orders of precedence and by varying the material condition modifiers on the datum feature references. Two of the possible tolerance specifications are shown in [Figure 6-64](#) to explain the difference between the RMB and MMB modifier application.

First, the effects of the RMB modifier are shown. No material boundary modifier is shown on the datum feature references, so RMB is assumed to apply on all of them. The keyseat has a position tolerance and datum feature reference that requires the keyseat position tolerance zone to be centered on and aligned with datum axis A. The keyseat width is identified as datum feature B. The hole has a position tolerance that is specified RFS with datum feature references to A primary RMB, B secondary RMB, and C tertiary.

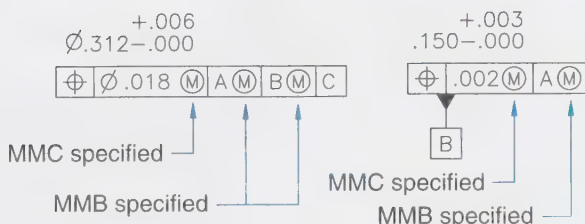


Datum feature references at RMB

Simulator contracts until the unrelated actual mating envelope is established

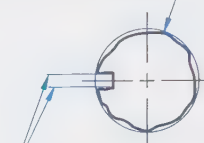


Simulator fixed in location expands from MMB until maximum possible contact is made (related actual mating envelope)



Datum feature references at MMB

Simulator fixed at MMB



Simulator fixed in location and fixed in size at the MMB of datum feature B

Goodheart-Willcox Publisher

Figure 6-64. The applicable material boundary modifier impacts the way in which the simulators establish the datums using features of size.

The simulators for the position tolerance on the hole are made up of the following: To establish the primary datum axis A, the smallest circumscribing cylinder is brought into contact with the primary datum A feature. Because the shaft is a primary datum feature, the simulator is a cylinder that is free to find a best fit to the datum feature. The simulator in this case is known as an *unrelated actual mating envelope*. It is called *unrelated* because it is not constrained (or related) to any higher precedence datums.

For secondary datum B, the simulator is an expanding pair of parallel planes that are centered on datum axis A. The planes expand equally while centered on primary datum axis A. They expand to make maximum possible contact with datum feature B while still remaining centered on and aligned with datum axis A. The planes form a *related actual mating envelope* for datum feature B. The envelope is considered *related* because it is constrained relative to a higher order of precedence datum.

In the lower portion of **Figure 6-64**, the effects of the MMB modifier are shown. When the datum feature references include the MMB modifier, the

location and orientation relationships between the simulators are unchanged. However, the simulators must be fixed in size to the appropriate MMB of the datum feature. The lower half of the figure shows the MMB modifier on the datum feature references. The simulators used to meet these requirements are fixed in size equal to the applicable MMB envelopes.

The simulator for the primary datum feature is equal to the maximum material size limit of the shaft, because only a size tolerance is applied to the primary datum feature. The secondary datum simulator is constrained (fixed in location and orientation) relative to the higher precedence primary datum feature simulator. The secondary datum simulator is sized to the .148" maximum material boundary that is created by the position tolerance on the keyseat. The part is free to move relative to the simulators by an amount equal to the datum feature variation from the MMB.

The concepts explained above are applicable to datum feature references made at LMB except that the simulators are established at the least material boundary instead of the maximum material boundary.

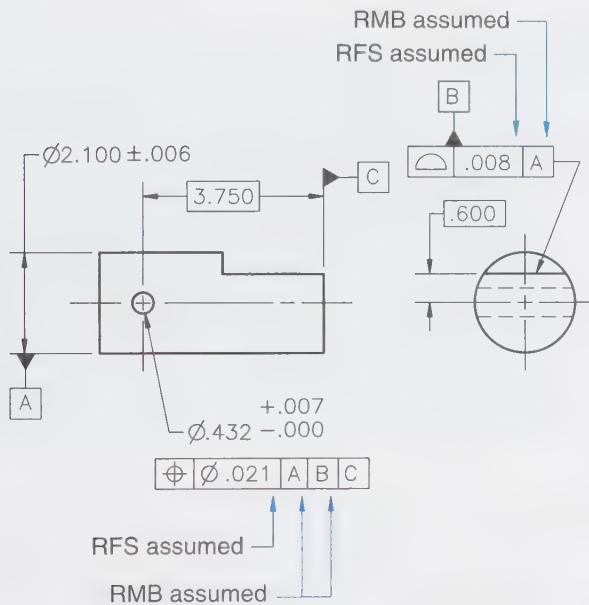
RMB, MMB, and LMB Application to Datum Feature Surfaces

Application of material boundary modifiers on a surface may be done only when material boundaries are created for the surface. Material boundaries may be created through the application of a profile tolerance. See Figure 6-65. A shaft with a flat is given. The diameter of the shaft is identified as datum feature A. The flat has a profile tolerance that is referenced to datum A. The flat is identified as datum feature B. There is a hole through the center of the shaft and parallel to the flat. The position tolerance on the hole references primary datum feature A, secondary datum feature B, and tertiary datum feature C. The figure shows the difference between referencing the primary and secondary datum features at RMB and MMB.

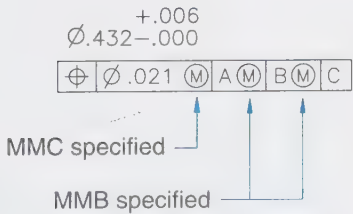
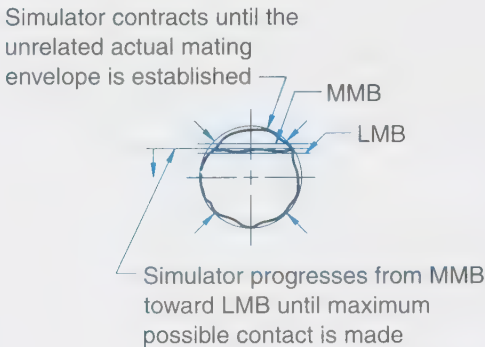
When the datum feature references are at RMB, the simulator for the primary datum is a

circumscribing cylinder contracted around the shaft until it makes full contact. The simulator for the flat surface is a plane parallel to datum axis A. This plane is fixed in its orientation to datum axis A and must also make contact with the surface. To bring it into contact with the feature, the simulator progresses from the MMB toward the LMB until it makes maximum possible contact. The simulator must remain within the allowable profile tolerance zone (between the two material boundaries).

Application of the MMB modifier on the primary datum feature reference requires the simulator to be a cylinder with a diameter equal to the MMB of the shaft. The MMB modifier on the secondary datum feature reference requires the simulator be a plane fixed in orientation and location relative to the primary simulator. The location of the secondary datum simulator is at the maximum material boundary created by the profile tolerance.



Datum feature references at RMB



Datum feature references at MMB

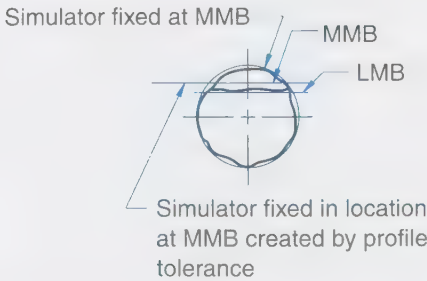


Figure 6-65. The applicable material boundary modifier impacts the way in which the simulators establish the datums using surfaces.

The application of the MMC (or MMB) modifier on a surface was not permitted prior to the ASME Y14.5-2009 standard. The application of MMB on a flat surface is a new concept based on material boundaries created by profile tolerances. It should be expected that the application of MMB on a surface will need to be explained to people that have training or experience based on previous dimensioning and tolerancing standards.

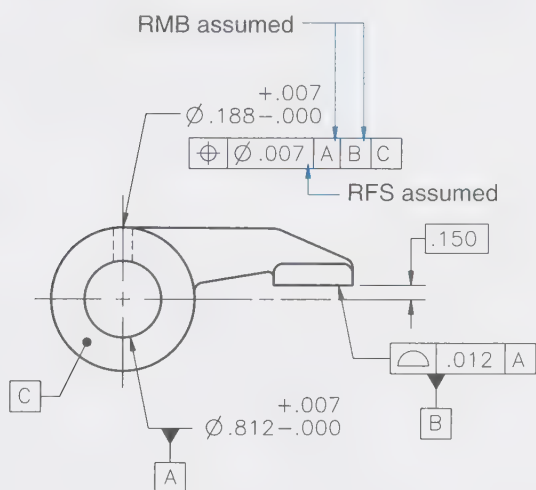
Material Boundary Modifiers on Offset Datum Features

A datum feature that is offset from a datum axis is sometimes utilized to stop the rotation of a part relative to higher precedence datums. The following explanation shows the significance of the material boundary modifiers and other notations

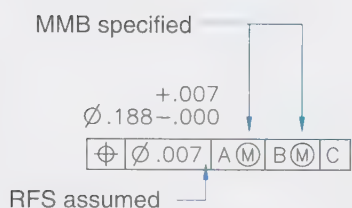
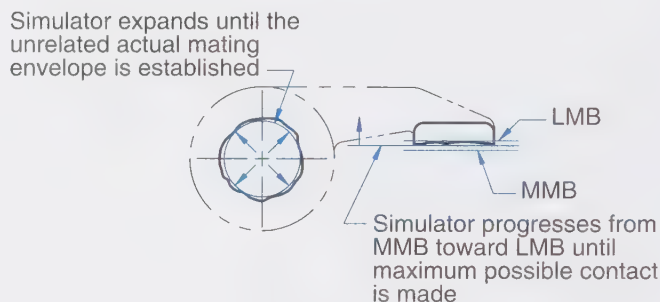
when they are applied to an offset datum feature.

Figure 6-66 shows a part that has a hole that is identified as the primary datum feature, and that establishes the primary datum axis. The secondary datum feature is a flat surface that is offset from the primary datum axis. This surface has a profile tolerance that is related to the higher precedence datum. A face surface perpendicular to the primary datum axis is identified as the tertiary datum feature. There is a small hole perpendicular to the primary datum axis that includes a position tolerance and appropriate datum feature references.

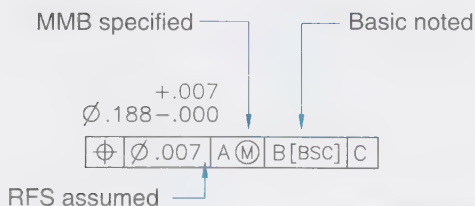
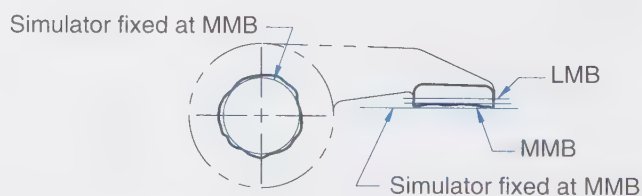
When the datum feature references are made RMB, the simulator for the primary datum is an inscribing cylinder that expands to make maximum possible contact with the hole. The simulator for the secondary datum feature is basically



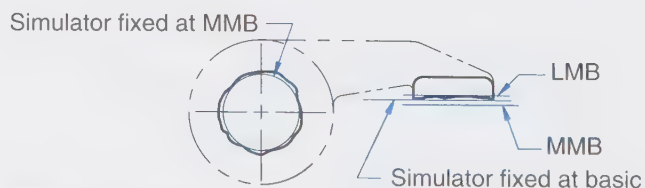
Datum feature references at RMB



Datum feature references at MMB



Datum feature references at basic



Goodheart-Willcox Publisher

Figure 6-66. The simulator for a surface that has material boundaries may be referenced at RMB, MMB, LMB, or at the basic location of the datum feature.

oriented to the higher precedence datum and it progresses from MMB toward LMB to make full contact with the surface. The simulator must remain within the profile tolerance for the datum feature.

When datum feature B is referenced at MMB, the simulator for the datum feature is fixed in location and orientation at the maximum material boundary created by the profile tolerance. It is also possible to use the LMB modifier on the datum feature reference, and in that case the simulator is theoretical and is fixed in location at the LMB.

If it is desired that the simulator be at the basic location of the feature, there are multiple means for specifying this. The word [BASIC] or abbreviation for basic [BSC] may be placed within brackets and shown in the feature control frame following the datum letter. It is also possible to use a drawing note to explain that the simulator is to be fixed at the basic location or the basic dimension value may be placed within brackets in place of the [BSC] notation following the datum feature letter.

Pro Tip

The notation of basic on a datum feature reference should be used with caution, especially in situations where the datum feature extends across a higher precedence datum axis as in **Figure 6-65**. For this type of application, the specification of basic could result in the part not fitting on the datum feature simulator when the datum feature has departed from the basic dimension.

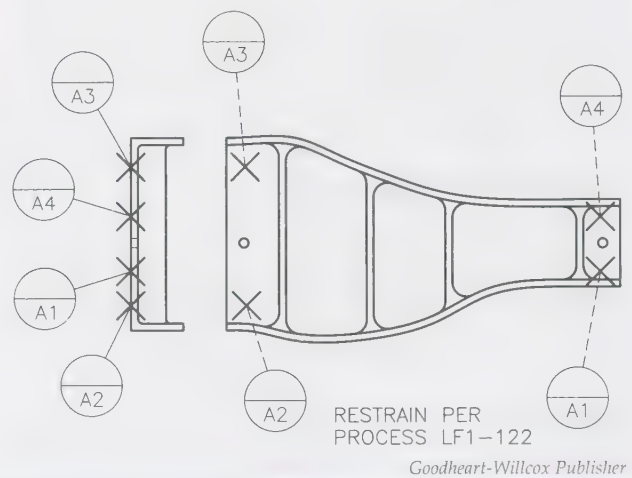
Restrained Datum Features

Tolerance requirements are applicable to parts in their free state condition. This is not typically a concern when parts are relatively rigid. However, when parts are subject to deflections that impact establishing a repeatable datum reference frame, it may be necessary to indicate an allowable restraining force. Some examples of this are automotive body panels and aircraft structural parts. But the need to specify a restraining force is not limited to large metal parts. Small parts and those made of plastic and other materials will, depending on size and geometry, at times need to have restraining forces defined. Caution is also necessary when

specifying allowable restraint forces, especially in structural applications. Application of excessive force could result in preloading parts in assembly and thus creating a negative effect on structural integrity and fatigue life.

See **Figure 6-67**. The given part has a thin web, and it is used in an assembly where fasteners will hold its shape. In the given figure, datum targets are applied. A primary datum feature for a rigid part could normally be identified with either three target points or target areas. Using three targets for a primary datum on a thin part or a large flexible part is not always adequate. Application of only three targets could result in distortion in the unsupported areas of the part. To prevent undesired distortion, or to pull distortion out of a part, additional datum targets may be used in combination with notation of restraint requirements.

A note may define the restraint conditions (amount of force and location) or reference a process document. Restraining parts to achieve requirements does introduce risks related to stress levels that may be induced into the parts. Caution should be exercised when noting allowable restraint if the parts are used in structural applications where preloads are detrimental to the design application. If not properly accounted for in the assembly processes, a part restrained for acceptance at the detail level could introduce variation into an assembly.



Goodheart-Willcox Publisher

Figure 6-67. A restraining requirement may be specified in the product definition when parts are subject to free state variation or when it is desirable to pull distortion out of a part.

Chapter Summary

- ✓ Datum feature references make it possible to define toleranced relationships to true geometric planes and axes.
- ✓ A datum feature is a physical feature on a part. Datum features are identified on drawings and on CAD solid models.
- ✓ Physical datum features are used to locate theoretical datum planes, axes (lines), and points.
- ✓ References shown in a feature control frame are made to the physical features that establish the theoretical datum planes and axes.
- ✓ Datum simulators (tools) are used to locate the theoretical datums from the physical locations of the datum features on a part or object.
- ✓ All datum feature references include material boundary modifiers, either implied or shown.
- ✓ The number of datum targets used is dependent on the order of precedence for the datum.
- ✓ Multiple feature control frames showing position or profile tolerances that reference the same datum reference frame establish a single pattern (simultaneous requirements) of toleranced features.
- ✓ A secondary or tertiary datum feature reference including an MMB modifier results in a requirement that the datum be simulated at its applicable maximum material boundary.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. ____ mutually perpendicular planes form a complete datum reference frame.
A. Two
B. Three
C. Four
D. Both A and B.
2. A datum reference frame may be thought of as the theoretical planes and axes that set up a(n) ____.
A. coordinate system
B. inspection system
C. tooling system
D. feature control frame
3. A flat datum surface is used to establish a datum ____.
A. point
B. line
C. area
D. plane
4. A datum axis may be established from a ____.
A. hole
B. shaft
C. centerline
D. Both A and B.
5. In a drawing view, the leader on a datum target symbol is made ____ if the target is on the far side of the feature.
A. dashed
B. solid
C. phantom
D. None of the above.
6. A ____ datum feature reference is always the first one shown in a feature control frame.
A. letter A
B. primary
C. secondary
D. tertiary
7. When a datum plane and datum axis are referenced in the same feature control frame, ____.
A. the plane must be the primary datum
B. the axis must be the primary datum
C. one of the two is selected as primary based on design function
D. None of the above.
8. Identifying a cylindrical feature of size as a datum feature establishes a datum ____.
A. axis
B. plane
C. target
D. symbol

9. Rule #2 of the dimensioning and tolerancing standard makes the RMB modifier applicable on datum feature references that are ____ unless shown otherwise.
 - A. surfaces
 - B. features of size
 - C. A and B
 - D. Neither A or B.
10. A primary datum feature reference including an MMB modifier requires the datum feature be simulated at its ____.
 - A. least material limit
 - B. maximum material boundary
 - C. actual mating envelope
 - D. actual produced size
11. When using datum target points to establish a primary datum plane, at least ____ points must be used.
 - A. one
 - B. two
 - C. three
 - D. six
12. Two datum letters separated by a dash within a single cell of a feature control frame indicates ____.
 - A. an error
 - B. primary and secondary datums
 - C. multiple (compound) datum features
 - D. all the letters have been used and double letters are now in use
13. A ____ dimension must be shown between offset surfaces that are used to establish a single datum plane.
 - A. basic
 - B. close toleranced
 - C. reference
 - D. None of the above.
18. *True or False?* The datum feature references in a feature control frame must be shown in alphabetical order.
19. *True or False?* A hole may be used as a datum feature of size, but this practice has no practical application.
20. *True or False?* In regard to datum references, based on Rule #2 of the dimensioning and tolerancing standard, datum feature references are assumed to apply RMB.
21. *True or False?* A reference to multiple (compound) datum features means the two features are used to establish two datums.
22. *True or False?* A datum feature symbol is generally *not* applied to a line representing a center plane.
23. *True or False?* The three datum target types are *not* to be combined to establish a single datum.

Fill in the Blank

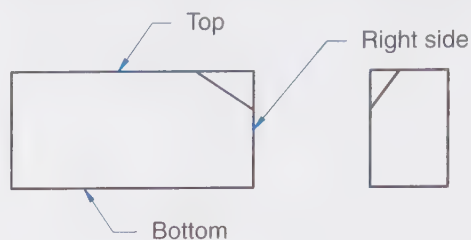
24. The intersections of the planes in the datum reference frame establish _____.
25. A _____ symbol is used to identify a feature from which a datum is established.
26. Datum targets may be _____, _____, or _____.
27. Datum _____ is determined by the order in which datum features are referenced in the feature control frame.
28. To identify a cylinder as a datum feature, a datum feature symbol is placed on the _____ dimension.
29. A datum feature that is a flat surface may be identified in a drawing view by placing the datum feature symbol on a(n) _____ line from the surface.
30. A datum feature symbol applied to the width dimension on a rectangular slot establishes a datum _____ at the center of the slot.
31. A secondary datum feature reference including an MMB modifier requires the datum feature be simulated at its _____ material boundary.
32. A minimum of _____ points are required to establish a secondary datum plane when using datum target points.

True/False

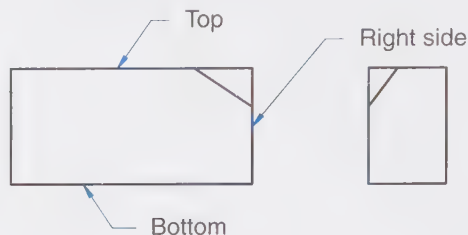
14. *True or False?* Regardless of the surface variations on a part, the planes forming the datum reference frame are always mutually perpendicular.
15. *True or False?* A datum axis is *not* identified by placing the datum feature symbol on a centerline.
16. *True or False?* A datum target area is outlined with a line made of short dashes.
17. *True or False?* The order of precedence for datums is defined by the order shown in the feature control frame.

33. A movable datum target symbol indicates that the target simulator must _____ to make contact with the part.

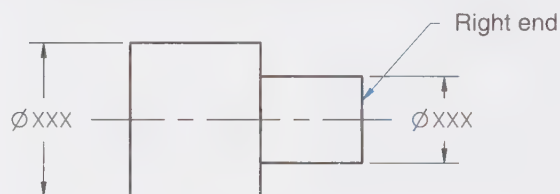
42. Identify the bottom surface on the given part as datum feature B.



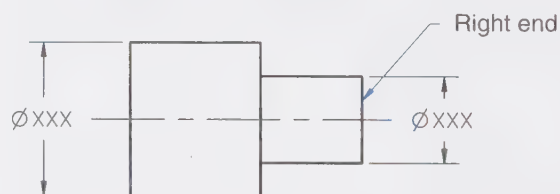
43. Identify the right side surface on the given part as datum feature C.



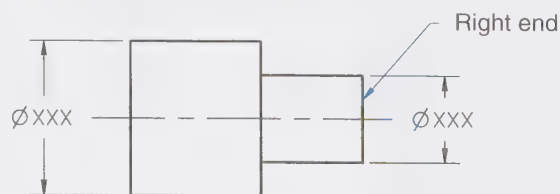
44. Identify the small diameter as datum feature A to establish a datum axis.



45. Identify the large diameter as datum feature B to establish a datum axis.



46. Identify the right end surface on the given shaft as datum feature C.



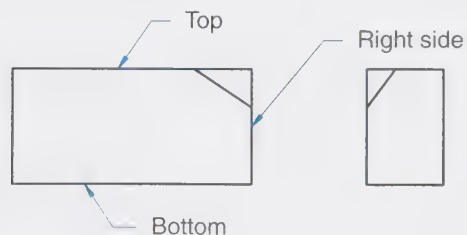
Short Answer

34. Explain the difference between a datum feature and a datum.
35. When basic dimensions are shown between datum targets, what is the tolerance on the location of those targets?
36. How can a datum plane be simulated when the datum feature is a flat external surface?
37. When is it necessary to show material boundary modifiers on datum feature references?
38. Explain how to establish a primary datum axis when a hole has been referenced at MMB.
39. The size of a datum target area should *not* be arbitrary. Explain how at least one consideration impacts the size of a target area.
40. Explain why the only two target points on a flat surface, labeled E1 and E2, should *not* be referenced as a primary datum.

Application Problems

Some of the following problems require that a sketch be made. All sketches should be neat and easily readable. Each problem description requires the addition of some dimensions or symbols for completion of the problem. Complete the actions defined in the instructions in a manner that is in compliance with dimensioning and tolerancing requirements.

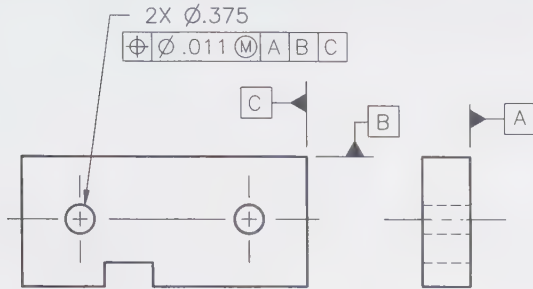
41. Identify the top surface on the given part as datum feature A.



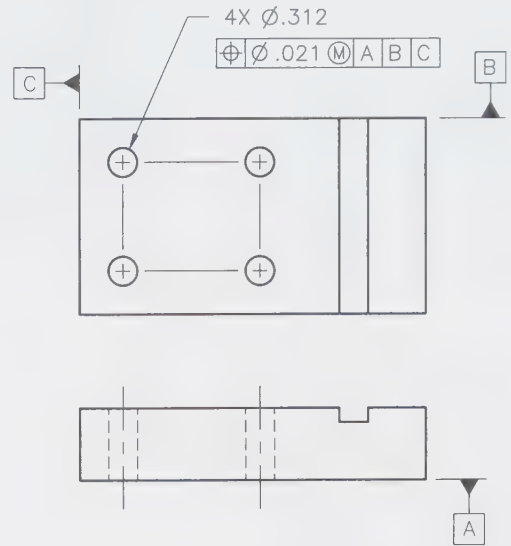
47. Label the primary, secondary, and tertiary datum feature references in the given feature control frame.



48. Sketch a tool to simulate the three required datum planes and show the part in the tool. Indicate the number of points that must be in contact with each datum simulator.

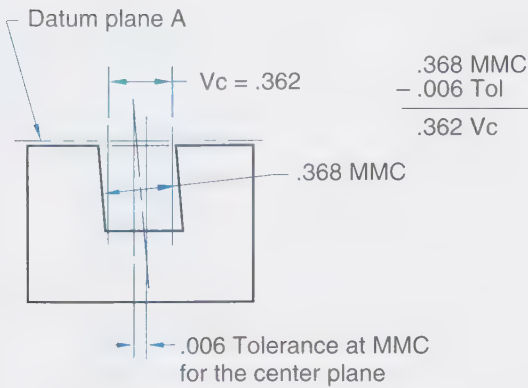


49. Revise the given drawing to include datum target points for all three required datums. Locate the targets in positions that you feel best stabilize the part and dimension their locations. Show target symbols to identify the targets.



$.375 \pm .007$
 $\perp .006 \text{ (M) A}$

Drawing requirements for a slot

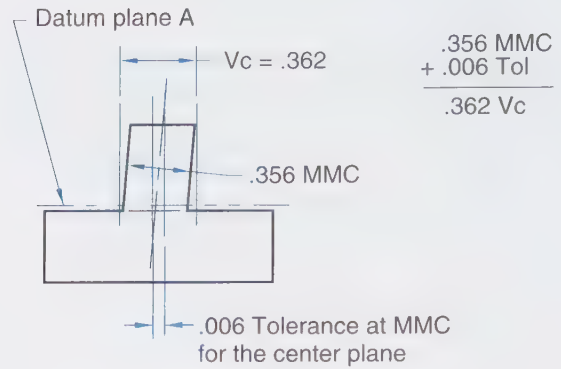


Effect of a perpendicularity tolerance on a slot

Orientation applied to an internal feature of size

$.349 \pm .007$
 $\perp .006 \text{ (M) A}$

Drawing requirements for a rail



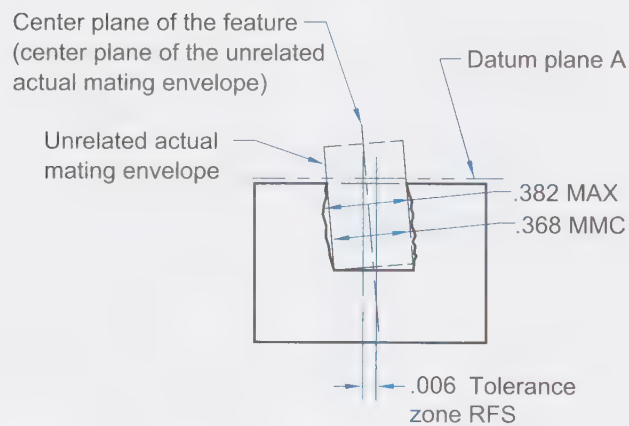
Effect of a perpendicularity tolerance on a rail

Orientation applied to an external feature of size

Goodheart-Willcox Publisher

$.375 \pm .007$
 $\perp .006 \text{ A}$

Drawing requirements for a slot



Orientation terminology

Goodheart-Willcox Publisher

Chapter 7

Orientation Tolerances

Objectives

Information in this chapter will enable you to:

- ▼ Identify, apply, and interpret orientation tolerances.
- ▼ Complete orientation tolerance specifications including one or two datum feature references.
- ▼ Explain the effects of material condition modifiers when orientation tolerances are applied to features of size.
- ▼ Calculate the virtual condition for internal and external features of size to which an orientation tolerance is applied.
- ▼ Complete tolerance specifications that include orientation and form requirements on a single feature.

Technical Terms

actual mating envelope
actual mating envelope axis
derived median line
orientation tolerances
related actual mating envelope
unrelated actual mating envelope
virtual condition

Introduction

Orientation tolerances are used to control the parallelism, perpendicularity, or angularity variation of a feature relative to one or more datums. These tolerances are specified using a feature control frame and may be applied to surfaces and to features of size. As with other tolerances, the method of application determines whether the

tolerance establishes a requirement on a surface or a feature of size.

Orientation tolerances are always referenced to datums, and therefore have a well-defined origin for completion of measurements. The exactness and clarity of requirements of orientation tolerances are not achievable through the utilization of general angle tolerances.

Orientations between features are sometimes controlled by an angle dimension that includes a “general angle tolerance” expressed as a plus or minus value. Angle tolerances applied in this manner do not always result in adequate definition of requirements. It is better to use orientation tolerances instead of the plus and minus tolerancing method.

Orientation Tolerance Categories

Parallelism, perpendicularity, and angularity are orientation tolerances. An orientation tolerance applied to a flat surface or to a feature of size such as a slot or rail produces a tolerance zone that is bounded by parallel planes. The planes forming the tolerance zone are in perfect orientation relative to the referenced datums. An orientation tolerance applied to a cylindrical feature of size produces a tolerance zone that is bounded by a cylinder. The cylinder forming the tolerance zone is in perfect orientation to the referenced datums. Whether a surface, center plane, or axis is contained within the tolerance boundary depends on the way in which the tolerance is applied and the type of feature to which it is applied.

Symbols and Application

There are three symbols that may be applied to define orientation tolerances. Those symbols are

angularity, parallelism, and perpendicularity. See **Figure 7-1**. It is an acceptable practice to apply the angularity symbol to a feature that is at any angle relative to the referenced datums. However, the parallelism and perpendicularity symbols may be used when features are parallel or perpendicular to the referenced datums.

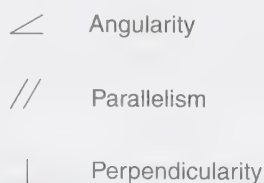
Traditional practices still utilize all three symbols, but it is an acceptable alternate practice to utilize only the angularity symbol for all orientation tolerances.

The practice of using only the angularity symbol may simplify tolerance application in orthographic views where standardized rules apply regarding lines shown perpendicular and parallel. Perpendicular lines are assumed to be at 90° to one another in orthographic views, and parallel lines are assumed to be parallel. Those assumed angles become basic when an orientation tolerance is applied.

In solid models, no angles are assumed. The features must be dimensioned or the angle queried on the model. Since no angle is assumed in a model, the application of a perpendicularity symbol for a tolerance on a surface will indicate the surface is at a 90° angle relative to the primary datum. No query is required because the application of the symbol clearly indicates the surface is at 90°.

If a query between the tolerated surface and the primary datum indicates an angle other than 90°, then the perpendicularity symbol should not be used. The same is true for parallel surfaces. When the parallelism tolerance is applied to a surface, it is known to be parallel to the referenced datum.

The angularity symbol is made of two lines, one drawn horizontal and the other at a 30° angle. Angularity is traditionally applied to features at any orientation other than parallel or perpendicular. Parallelism is indicated by two parallel lines drawn at a 60° angle from horizontal. Parallelism is only used to indicate a tolerance on features that are parallel to the referenced primary datum. Perpendicularity is defined by two perpendicular



Goodheart-Willcox Publisher

Figure 7-1. There are three orientation tolerance symbols.

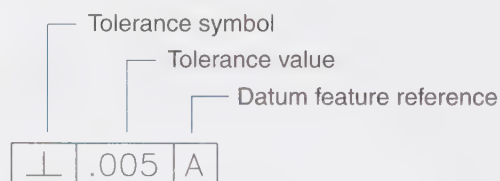
lines. Perpendicularity is only used on features that are at a 90° angle relative to the referenced primary datum.

Feature Control Frames

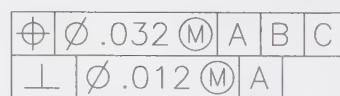
Feature control frames for orientation tolerances are generally drawn in the same format as those for other tolerances. See **Figure 7-2**. The tolerance symbol (characteristic) is shown on the left, followed by the tolerance value, and then the datum feature references. An orientation tolerance specification must include at least one datum feature reference. The first example in **Figure 7-2** shows an orientation (perpendicularity) tolerance of .005" relative to datum A.

Orientation tolerances may be combined with other tolerances in a multiple-segment (multiple-line) feature control frame. The second example in **Figure 7-2** shows a two-segment (two-line) feature control frame in which an orientation tolerance is used to refine a position tolerance. The given example has a diameter .032" position tolerance and a diameter .012" orientation tolerance.

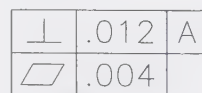
Although position tolerances have not yet been discussed, a position tolerance controls both location and orientation. If orientation is more critical than location, then an orientation tolerance may be used to refine the control provided



Orientation tolerance



Orientation as a refinement of position



Form as a refinement of orientation

Goodheart-Willcox Publisher

Figure 7-2. Orientation tolerances may be used to refine positional tolerance requirements. Form tolerances may be used to refine orientation tolerance requirements.

by the position tolerance. This permits the position tolerance to be maximized while specifying the orientation to a smaller value. When an orientation tolerance is used in combination with a position tolerance, the orientation tolerance must be smaller than the position tolerance.

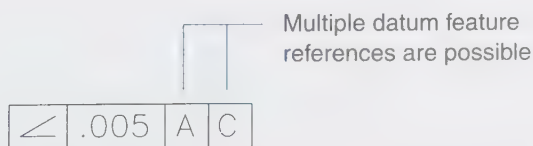
Orientation tolerances applied to a flat surface may not adequately control the form of the feature. A form tolerance such as flatness or straightness may be used to further refine the feature requirements within the allowable orientation tolerance. The third example in **Figure 7-2** shows a form tolerance of .004" that is smaller than the orientation tolerance of .012". When an orientation tolerance is applied in combination with a form tolerance on a surface, the form tolerance must be less than the orientation tolerance. This requirement is established in the ASME Y14.5-2009 standard.

Multiple-segment feature control frames are limited only by the needed control for the feature. Multiple-segment feature control frames could look overly restrictive when first seen. Actually, they allow each of the tolerance values to be maximized where properly used.

Using a single feature control frame to achieve both the orientation and flatness control shown in the third example in **Figure 7-2** would require an orientation tolerance of .004". That would mean that the surface must be flat and perpendicular within .004". An orientation tolerance of .004" is significantly smaller than .012" and could result in a cost increase. The given two-segment feature control frame permits the maximum amount of tolerance for each of the two controls. It permits orientation to vary within .012" and requires the feature be flat within .004".

Orientation tolerances must reference at least one datum feature. See **Figure 7-3**. There are applications where it is necessary to reference two datum features to adequately constrain the degrees of freedom.

A reference to a single datum feature establishes orientation to only one datum. There is no implied relationship to any other feature. When an orientation tolerance references two datum



Goodheart-Willcox Publisher

Figure 7-3. Orientation tolerances may include more than one datum feature reference.

features, the primary datum is the one from which the required angle is measured, and the secondary datum establishes a fixed rotation of the part on the primary datum. This is illustrated later in this chapter.

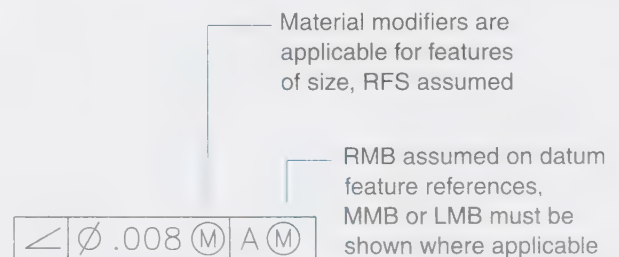
Application to Surfaces and Features of Size

Orientation tolerances may be applied on surfaces and features of size. When applied to surfaces, the tolerance controls only the indicated surface and the maximum allowable variation is shown in the feature control frame. Material condition modifiers are not applicable on the orientation tolerances when they are applied to a surface.

When applied to a feature of size, there is always an applicable material condition. The extent of orientation control established by the tolerance depends on the applicable material condition modifier. Regardless of feature size (RFS) application establishes a control of the center plane or axis, and the maximum allowable variation is defined by the value shown in the feature control frame. Maximum material condition (MMC) application also controls the center plane or axis, the allowable tolerance is equal to the value in the feature control frame, and that tolerance value increases when the feature departs from the MMC size. Additionally, MMC application creates a virtual condition boundary that the surface of the feature cannot violate. This will be further explained.

Material Condition and Material Boundary Modifiers

Tolerances applied on features of size always have an applicable material condition. The applicable modifier is either implied or explicitly shown. See **Figure 7-4**. All tolerances, including orientation tolerances, are assumed to apply regardless of feature size (RFS) unless a modifier is shown. Modifier symbols that may be shown are maximum material condition (MMC) and least material condition (LMC). The MMC and LMC symbols are placed following the tolerance value. There is no



Goodheart-Willcox Publisher

Figure 7-4. A material condition modifier is applicable to orientation tolerances on features of size.

symbol for RFS because it is the default applicable condition and is understood to be applicable when neither the MMC nor LMC modifier is shown.

Datum feature references always have an applicable material boundary. The applicable modifier is either implied or explicitly shown. All datum feature references, including those on orientation tolerances, are assumed to apply regardless of material boundary (RMB) unless a modifier is shown. When datum features have a maximum and minimum material boundary, the MMB or LMB symbol may be applied. The MMB and LMB symbols are placed following the datum feature reference letter. There is no RMB symbol because it is the default boundary condition and is understood to be applicable when neither the MMB nor LMB modifier is shown.

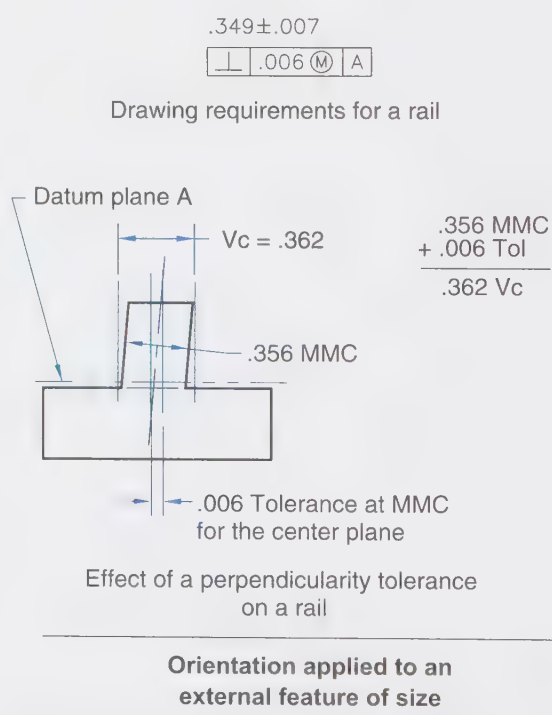
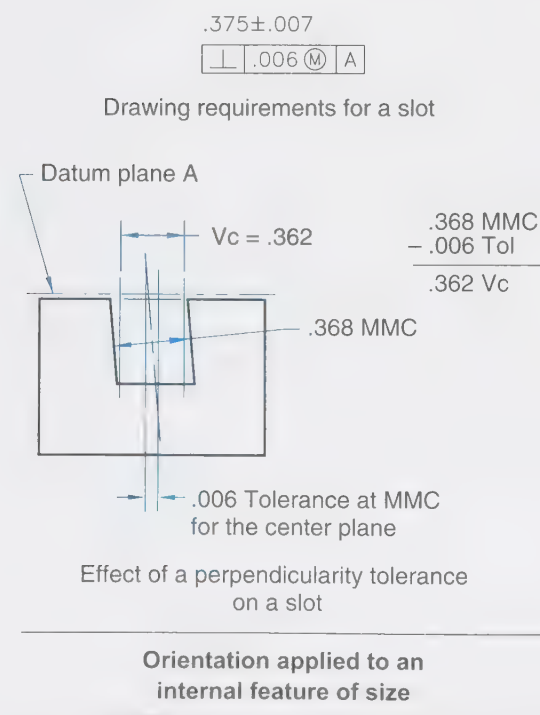
Virtual Condition (MMC Application)

When an orientation tolerance is applied to a feature of size and includes an MMC modifier, a virtual condition is established. See Figure 7-5. The *virtual condition* is a boundary that cannot be violated by the surface of the tolerated feature. Its size is a result of the combined effect of the specified orientation tolerance and the MMC size of the feature. When the virtual conditions of two mating features are equal or provide clearance, the two features will freely fit together. In the given

figure, the slot and rail both have a virtual condition of .302" that is created relative to a common interface datum, so it is known that the rail will fit into the slot.

An orientation tolerance applied to an internal feature such as a slot creates a virtual condition that is smaller than the MMC size. See Figure 7-5. As the allowable orientation variation on an internal feature increases, the apparent size (mating envelope) of the opening decreases. In other words, the usable width of the slot gets smaller. The given example shows how a slot appears to get smaller as its orientation departs from perpendicular. Vertical blue lines in the figure show the virtual condition and the tolerance zone for the center plane. The surface is not allowed to violate the virtual condition, and the center plane is not allowed to violate the tolerance zone. When an internal feature has imperfections, the center plane is located at the center of the actual mating envelope. The *actual mating envelope* is a perfect shape that is expanded inside the imperfect internal feature to make contact.

The virtual condition of an internal feature is calculated by subtracting the orientation tolerance from the MMC size. The virtual condition is, in effect, an envelope inside of which no segment of the internal feature may extend. No part of the side surfaces on the shown slot extend inside the



Goodheart-Willcox Publisher

Figure 7-5. A virtual condition is created when an orientation tolerance includes an MMC modifier and is applied to a feature of size.

virtual condition boundary. The virtual condition acts like a tolerance boundary for the surface. It should also be noted that the figure shows the tolerance zone for the center plane. The center plane in the given figure is within the tolerance zone.

An orientation tolerance applied to an external feature such as a rail creates a virtual condition that is larger than the maximum material condition size. See **Figure 7-5**. The given example shows how a rail appears to get larger as its orientation departs from perpendicular. Vertical blue lines in the figure show the virtual condition and the tolerance zone for the center plane. The surface is not allowed to violate the virtual condition, and the center plane is not allowed to violate the tolerance zone. When a feature has imperfections, the center plane is located at the center of the actual mating envelope. The actual mating envelope for an external feature is a perfect shape that is contracted on the imperfect feature to make contact.

The virtual condition of an external feature is calculated by adding the orientation tolerance to the MMC size. The virtual condition is, in effect, an envelope outside of which no segment of the external feature may extend. No part of the side surfaces on the shown rail extend outside the virtual condition boundary. The center plane is also within the required tolerance zone.

The virtual condition boundary is a fixed size. It is not affected by the actual produced size of the feature. As the produced feature departs from its MMC size, the allowable orientation variation may increase without violating the virtual condition boundary. This allowable additional variation is often referred to as bonus tolerance, or additional tolerance.

The shown width of the tolerance zone is applicable when the feature is produced at the MMC size. As the feature departs from its MMC size, this tolerance zone expands to allow additional variation of the center plane. The increase in the width of the tolerance zone is equal to the feature's actual mating envelope departure from the MMC size.

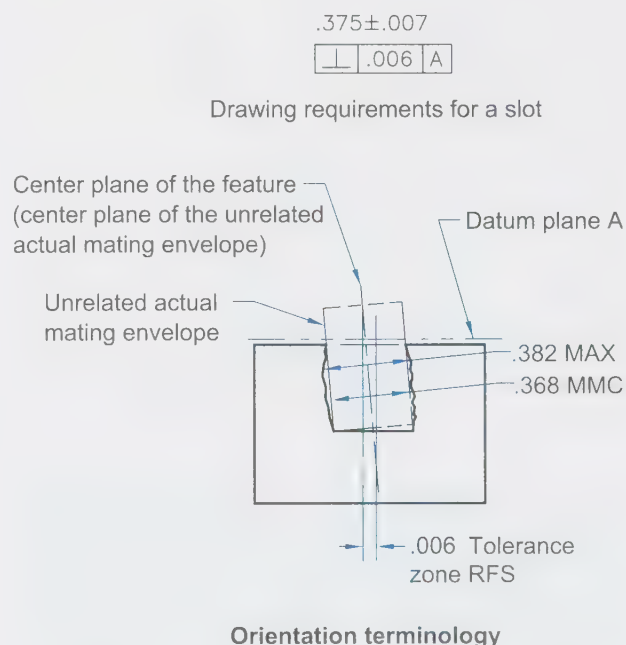
Feature Center Plane and Feature Axis

A center plane or axis is established in the same way for MMC and RFS applications. The tolerance zone for the center plane or axis may be variable in size or a fixed value depending on whether it is specified at MMC or RFS. If applied MMC, the tolerance zone may increase in size as the feature mating envelope departs from the MMC size. If applied RFS, the tolerance zone is a constant size.

An orientation tolerance applied to a feature of size establishes a tolerance zone inside of which the feature center plane or the feature axis must lie. No virtual condition is created when the orientation tolerance is applied RFS, but there is a worst case boundary.

In the case of a slot or rail, the feature center plane is a theoretically exact plane established at the center of the *unrelated actual mating envelope* for the slot or rail. See **Figure 7-6**. The actual mating envelope is considered *unrelated* because it is not constrained (or related) to the referenced datums. It is free to move to find the best fit to the feature. The feature center plane exists at the center of the actual mating envelope. The plane in the given figure is inclined (has orientation variation) because the actual mating envelope is inclined for a best fit to the produced slot. Establishing the center plane by using the unrelated actual mating envelope is applicable to RFS, MMC, and LMC applications.

The actual mating envelope must be no smaller than the MMC size of the slot. Also, two point measurements across the slot must be no larger than the maximum size limit. In RFS applications, the sides of the slot may have form variation up to the maximum allowed by the size tolerance or applied form tolerances. In MMC applications, the virtual condition may limit the allowable form variation.



Goodheart-Willcox Publisher

Figure 7-6. The center plane of a feature is defined as the center plane of the unrelated actual mating envelope.

In the ASME standard, the center plane of the slot is not the same as the derived median plane for the slot. A derived median plane is an abstract imperfect feature that follows the mid-points between the surface variations as described in Chapter 5. The derived median plane is only applicable to form tolerances. Orientation tolerances applied on an RFS basis are not defined by the standard as controlling the derived median plane. They instead control the center plane, which is established from an unrelated actual mating envelope.

A feature axis for a cylindrical feature is a theoretically straight line that is at the center of the unrelated actual mating envelope of the controlled cylindrical feature. The feature axis is not the same as the *derived median line* for a produced cylinder. Orientation tolerances on a feature of size apply to the *actual mating envelope axis* and not to the derived median line. Form tolerances on a feature of size control the derived median line and not the feature axis.

Parallelism

Two features drawn parallel in an orthographic drawing are assumed to have an implied angle of 0° between them unless dimensions are given to specify otherwise. The tolerances on these parallel and undimensioned features may in some instances be defined by a general angular tolerance, limits of size, or parallelism tolerances. Parallel features in an undimensioned solid model are not assumed to be parallel. The features must be queried to determine the angle between them. The angular relationships between parallel features in an orthographic drawing or a solid model may be toleranced with parallelism tolerances. Using a parallelism tolerance does not preclude showing additional requirements such as location tolerances. When other tolerances, such as a position tolerance, are applied on parallel features, it is also possible to include a parallelism tolerance to refine the angular relationship requirements.

Note

A parallelism tolerance defines the allowable orientation variation between the tolerated feature and referenced datum feature but does not affect location or size of the features.

Control of parallel features through a general note defining angular tolerance is somewhat ambiguous because it is not known which of any two parallel surfaces acts as the origin (or datum). The requirement created by a general angular tolerance can become more ambiguous as the number of parallel features increases. As an example, several slots on one surface are parallel to each other. Does the angular tolerance accumulate between each slot, or is each tolerance measured relative to some implied reference feature? This ambiguity may be avoided through the identification of appropriate datums and the application of parallelism tolerances using the needed number of feature control frames.

The same ambiguity exists if parallel features are dimensioned with plus and minus tolerances for location and size control. If the limits on the dimensions are used to control orientation, an accumulation of tolerances can occur. Careful dimensioning and the utilization of the origin symbol can reduce the ambiguity, but there is still some potential for confusion. Typically, the clearest means for tolerancing the angular relationship between parallel features is to identify an appropriate datum feature and use orientation tolerances that reference that datum feature.

Parallelism tolerances permit the orientation requirements between specific features to be defined without any accumulation of tolerances. They have the necessary attributes to specify the required level of control for parallelism without restricting size or location tolerances. Because the parallelism requirements may be completely separated from size and location tolerances, a significant advantage is achievable by avoiding overly restrictive size and location requirements.

The 0° angle between parallel features in an orthographic view is assumed to be a basic dimension when a parallelism tolerance is applied. In a solid model, no angles are assumed, but the application of a parallelism tolerance may only be used on features that are parallel. So, the application of a parallelism tolerance communicates that the angle is required to be 0° and basic. Any other angle would indicate an error in creation of the surface or in the application of the parallelism symbol.

Parallelism Applied to a Flat Surface

Orientation requirements between a flat surface and a datum plane may be specified with a parallelism tolerance. See [Figure 7-7](#). A feature control frame is used to specify the parallelism tolerance and it is attached to each surface that is to

Figure 7-7 shows a part on which the parallelism tolerance of .009" is less than the size tolerance of .020". The parallelism tolerance controls only the accuracy of the surface orientation and form. It does not affect the limits of the size tolerance,



The given figure shows that a perfect form boundary is established at the .448" maximum material size. No portion of the surface may protrude outside this boundary. Datum plane A is

located by the bottom surface of the part. The three high points on the datum feature surface will locate the datum plane. The parallelism tolerance zone is bounded by two planes separated by .009". These planes must be parallel to datum A and may be at any location so long as the surface of the part does not violate the size tolerance.

The part is acceptable if the upper surface remains within the perfect form boundary and also remains within the parallelism tolerance zone. The tolerance zone is permitted to float (translate relative to the datums) to any location, provided the zone remains constrained in rotation (remains parallel) to the referenced datums, and the minimum size limit is also maintained.

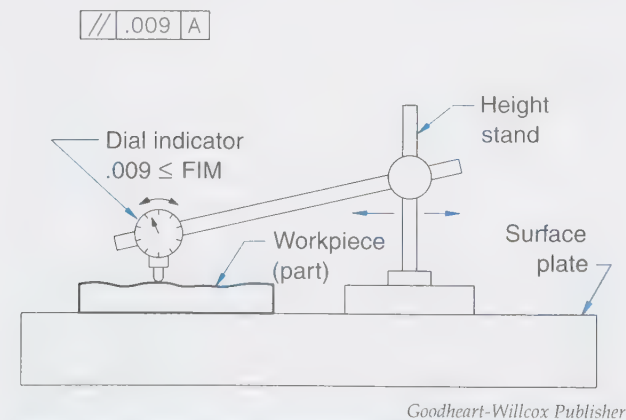
The minimum size limit is verified by direct measurements across the feature. Minimum size measurements are not made from the datum plane, but rather from the part surface.

Verification of Parallelism on a Flat Surface

One method for verifying parallelism of a flat surface is to use a dial indicator on a height stand and surface plate. See Figure 7-8. A surface plate is used to simulate datum A. The workpiece is set on the surface plate to establish the datum plane.

A height stand with a dial indicator is moved across the controlled surface. The dial indicator is kept at a fixed height as it is moved across the entire surface area. If the full indicator movement is less than or equal to .009", the parallelism requirement is met.

This method for verifying parallelism only checks the orientation requirement. It does not check the size tolerance. Before the part may be accepted, size tolerances must also be verified.

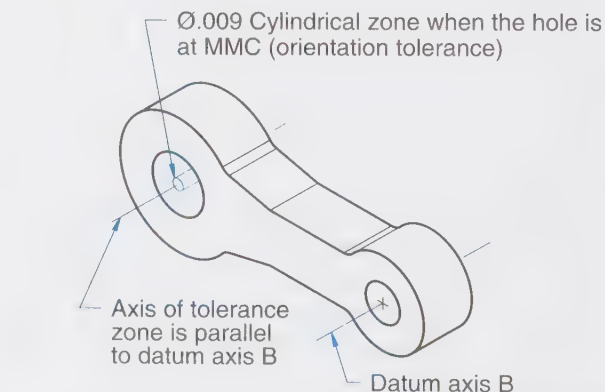
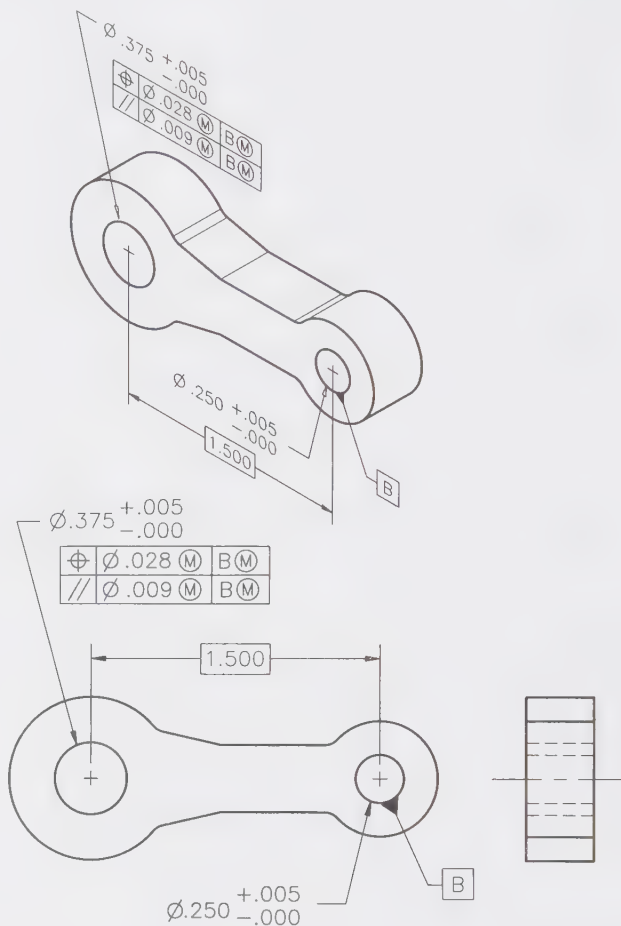


Goodheart-Willcox Publisher

Figure 7-8. Parallelism establishes only an orientation requirement for the feature. It does not control location.

Parallelism Applied to a Cylindrical Feature of Size

Parallelism may be used to control the orientation of a feature axis relative to either a datum axis or a datum plane. See Figure 7-9. The figure shows a parallelism tolerance used to control the axis orientation accuracy of one hole relative to the datum axis established by another hole.



Goodheart-Willcox Publisher

Figure 7-9. Parallelism may be used to establish an orientation requirement for one axis relative to another.

The given linkage has two holes. The .250" diameter hole is identified as datum feature B. The .375" diameter hole has a parallelism tolerance of diameter .009" at MMC relative to datum B at MMB. There is a .028" diameter position tolerance at MMC applied to that hole. Position tolerances have not yet been explained, so for now it is enough to say that this position tolerance specifies the allowable location variation for the hole. In addition to the orientation and position requirements, both holes also have size tolerances.

The orientation tolerance is a separate requirement from the position tolerance. Both must be met. The position tolerance applies to the hole. It does not apply to the orientation tolerance zone. The orientation tolerance zone has no required location. It is not constrained in translation relative to the referenced datum, it is only constrained in rotation relative to the referenced datum.

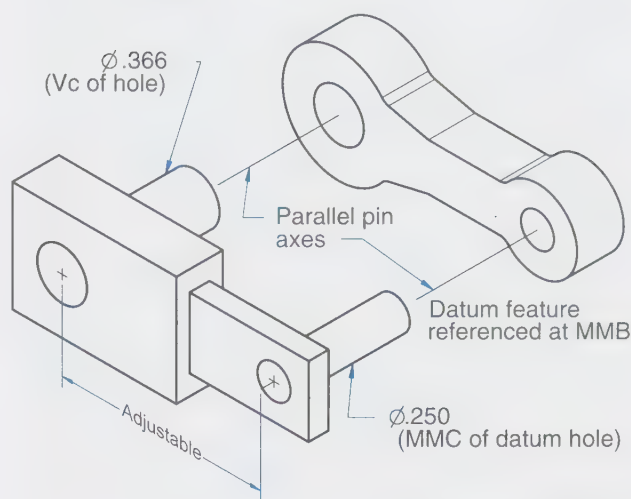
The datum axis is established by the hole identified as datum feature B. When the datum feature is at MMB, the datum axis must be centered on the hole. The parallelism tolerance zone is a .009" diameter cylinder that extends the length of the hole when the .375" hole is at MMC. The tolerance zone cylinder is parallel to the datum axis (constrained in two rotational degrees of freedom). Because the MMC modifier is shown on the tolerance value and the MMB modifier on the datum feature reference, additional tolerance is permitted if the controlled hole departs from MMC or the datum feature departs from MMB.

Verification of Parallelism Applied to a Feature of Size

Functional gages can typically be used to check orientation tolerances applied to features of size when the tolerances include the MMC modifier, but functional gages are not required. Other measurement processes may be used. Whether gages or other measurement methods are used, careful attention must be given to the proper gage design or measurement methods so as not to force unintended controls on the workpiece. As an example, the previously shown linkage must be checked for parallelism within .009" diameter while permitting hole location to vary within the .028" diameter position tolerance.

One possible functional gage for this linkage is shown in **Figure 7-10**. The gage has two parallel pins that are mounted on an adjustable slide. These pins simulate the MMB of the datum feature and the virtual condition created by the orientation tolerance. The slide permits the distance between the pins to be varied, but the pin parallelism is not

$$\begin{aligned} V_c &= \text{MMC} - \text{Tolerance} \\ &= .375 - .009 \\ V_c &= .366 \end{aligned}$$



Goodheart-Willcox Publisher

Figure 7-10. Gages used to verify parallelism must not force unrequired location requirements.

affected. This gage verifies the parallelism tolerance and does not verify the position tolerance.

The following two paragraphs explain how the pin diameters are determined. The gage pins have fixed diameters because the parallelism tolerance has an MMC modifier and the datum feature reference includes an MMB modifier. Values calculated are the theoretical simulator values.

Datum B is simulated by a pin equal in diameter to the MMB of the datum feature (the .250" hole). On the given linkage, the applicable MMB is equal to the maximum material condition of the datum hole, which is .250" diameter. The .250" diameter is the smallest permitted hole size, and by definition the smallest hole is known as the MMC size.

Establishing a datum axis is explained in detail in Chapter 6. Briefly, a datum feature of size referenced at MMB is simulated at the MMC size if the following conditions are met: the datum feature reference is a primary datum, the MMB modifier is applied to the datum feature reference, and no derived median line straightness or center plane flatness tolerance is applied to the datum feature.

The second pin on the gage has a diameter equal to the virtual condition, created by the parallelism tolerance, of the controlled hole. Any orientation tolerance applied on a feature of size may be checked by a gage that is sized to the feature's virtual condition if the tolerance value includes an MMC modifier. For a hole, the virtual condition is determined by subtracting the parallelism

tolerance from the MMC size. On the given part, the virtual condition of the hole is equal to .375" diameter (MMC) minus .009" (the parallelism tolerance). The virtual condition is .366" diameter.

The pins on the gage can be inserted into the holes if the linkage is made in compliance with the size limits and the parallelism tolerance. When both holes on the linkage are at MMC, the pins will fit into the workpiece if the parallelism variation is no more than .009".

The effects of the MMC and MMB modifiers can be seen through the examination of what happens when the gage in [Figure 7-10](#) is used on several different linkages. Consider a linkage that is produced with both holes at MMC. The datum B hole is at .250" diameter, and the other hole has a .375" diameter. When the gage is inserted into this part, the datum B simulator aligns with the .250" diameter hole. The .366" diameter pin is .009" smaller than the .375" diameter hole, and fits into it, provided there is no more than .009" parallelism variation.

The second example linkage is produced with the datum B hole at MMC, and the other hole has departed from MMC. The datum B hole has a .250" diameter. The second hole has a .380" diameter. The .380" diameter is a .005" departure from the MMC size of .375". When the gage is inserted into this part, the datum B simulator aligns with the datum axis. The .366" diameter pin is .014" smaller than the produced hole, and fits into it, provided there is no more than a .014" parallelism variation. *The departure of a feature from its MMC size results in an allowable increased tolerance when the MMC modifier is applied to the tolerance value.*

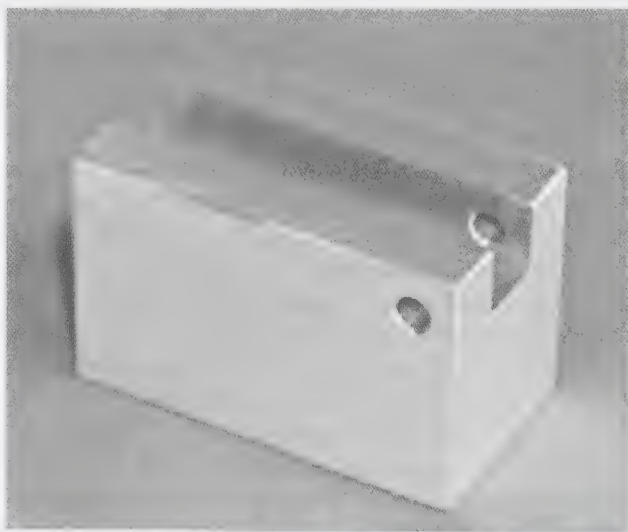
The third linkage to consider has both holes at a size larger than their specified maximum material condition. The datum B hole has a .255" diameter, and the other hole has a .380" diameter. When the gage is inserted into the part, the datum B simulator (pin) has clearance and is free to move (translate and rotate) within the datum B hole. The movement of the datum B simulator within the datum B hole increases the amount of allowable parallelism tolerance. The toleranced hole has additional clearance relative to the gage pin because of that hole departing from MMC, and that clearance permits additional orientation variation. The increased diameter of both holes impacts the allowable variation because the tolerance includes the MMC modifier and the datum feature reference includes the MMB modifier.

Perpendicularity

Perpendicularity tolerances are used to specify the orientation requirements of a feature that is at a 90° angle relative to a datum plane or axis. Perpendicularity tolerances may be applied to surfaces and features of size. See [Figure 7-11](#). The shown part contains multiple perpendicular surfaces and features of size that potentially could be toleranced with perpendicularity tolerances. Orientation tolerances for features that are 90° from the referenced datums are specified using the perpendicularity symbol. As an alternate practice, it is permissible to use the angularity symbol. The symbol used for 90° angles should be consistent within a single drawing or solid model. If the angularity symbol is used for 90° angles, then the perpendicularity symbol should not be used on the same drawing.

Drawing conventions do not require application of 90° angle dimensions on orthographic views. See [Figure 7-12](#). Any feature drawn to appear at 90° is assumed to be perpendicular unless an angle dimension is applied to indicate otherwise.

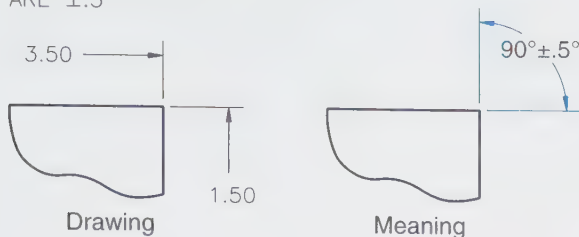
A 90° angle is not assumed in undimensioned solid models (3D models). The angle between surfaces must be determined by querying the model. However, if a perpendicularity tolerance is applied in a solid model, it is known that the query between the feature and datum should return a 90° angle.



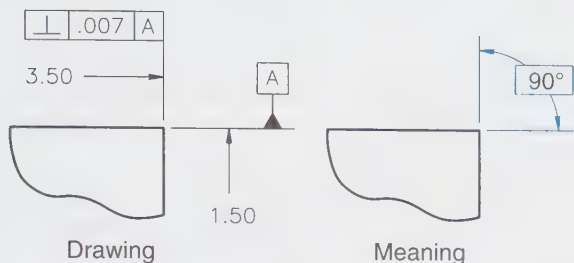
Goodheart-Willcox Publisher

Figure 7-11. Surfaces and features of size may require perpendicularity tolerances to control interrelationships.

NOTE:
ALL UNTOLERANCED ANGLES
ARE $\pm 5^\circ$



Implied 90° angle with bilateral tolerance



Implied basic 90° angle with perpendicularity tolerance

Goodheart-Willcox Publisher

Figure 7-12. Implied 90° angles are assumed to be basic when a perpendicularity tolerance is applied.

No assumption is made as to the allowable tolerances for 90° angles whether they are implied or dimensioned. The tolerance must be specified in some manner. The following are methods for specifying the allowable variation on perpendicular surfaces.

A general angular tolerance may be shown in the title block or in a general note. A note may also be used to relate all dimensions to a datum reference frame, thereby establishing perfect form and interrelationships at MMC, but this practice is not recommended. A unilateral or bilateral angle tolerance may be shown following the angle dimension value. Perpendicularity tolerances may be shown through the application of feature control frames.

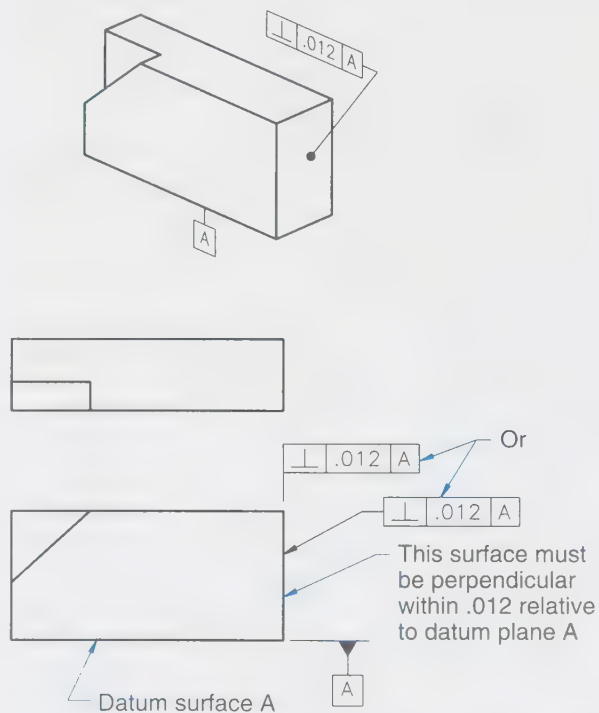
A general angular tolerance in the title block is assumed to apply to 90° angles if no note, directly applied tolerance, or feature control frame is applied to specify the orientation requirement. If a feature control frame is applied to control the perpendicularity, then the 90° angle is implied to be a basic dimension. It is not necessary to draw the 90° dimension to show that it is basic. If a perpendicularity tolerance is specified on a solid model, then the basic 90° between the toleranced surface and the datum is implied.

Perpendicularity Applied to Flat Surfaces

A perpendicularity tolerance is applied to a surface in an orthographic view by placing the feature control frame on an extension line or on a leader to the surface. See **Figure 7-13**. The tolerance must reference at least one datum. No material condition modifier is applied to the tolerance value when the controlled feature is a surface. In a solid model, the tolerance may be applied with a leader that terminates in a dot on the controlled surface. Feature control frames are not placed on extension lines in solid models. Placement on an extension line from a corner would not clearly indicate the surface to which the tolerance applies.

The given figure shows a perpendicularity tolerance applied to the right end of the given part. The tolerance only affects the end of the part to which it is attached. Because the shown tolerance is applied to the right end of the part, it has no direct effect on the left end, top, bottom, front, or back.

The shown perpendicularity tolerance references datum feature A. The bottom surface is identified as datum feature A. The tolerance



Goodheart-Willcox Publisher

Figure 7-13. Perpendicularity may be specified on a surface by placing the feature control frame on an extension line or leader.

specification and identification of the datum feature establishes an orientation control between the indicated surface and the referenced datum plane.

Referenced to One Datum Feature

A perpendicularity tolerance must include one or more datum feature references. A single datum feature reference is adequate for many applications. However, a reference to one datum feature constrains only the rotational degrees of freedom relative to the indicated datum. There are no assumed relationships to other datums.

Figure 7-14 illustrates a perpendicularity tolerance that references a single datum feature. The feature control frame is applied to the right side of the part in the given figure. It references datum feature A, and the bottom surface of the part is identified as datum feature A.

The length of the part is controlled by a size dimension of $2.00 \pm .03$ ". The perpendicularity tolerance is .005". The size tolerance and perpendicularity tolerance are separate requirements.

There is a perfect form envelope for the two ends at the MMC dimension of 2.03". This perfect form envelope controls only parallelism of the two ends; it does not control perpendicularity of the end surface relative to datum A. Based on only the size dimension, the two ends may have any shape or orientation, provided the ends do not violate the perfect form envelope or the minimum size dimension.

The right end of the part is controlled by more than the 2.00" size dimension. The shown perpendicularity tolerance establishes allowable orientation variation of the right end. This tolerance requires the surface fall between two parallel planes separated by .005". The two planes must be perpendicular to datum plane A. Datum plane A is located by the high points on datum surface A.

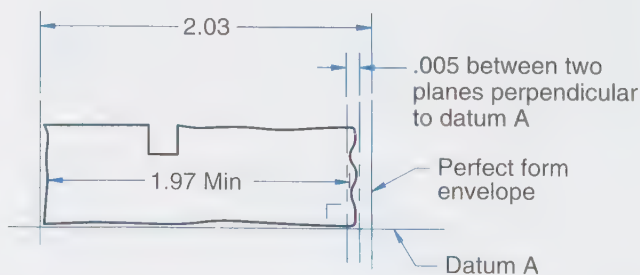
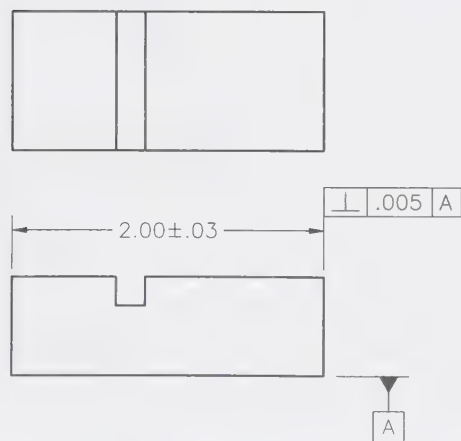
Perpendicularity of a surface may be verified using any number of methods including the one shown in **Figure 7-15**. An angle block is set on a surface plate. The workpiece is clamped with datum feature A against the angle block.

A height stand is positioned to hold a dial indicator in contact with the controlled surface. The height stand is moved across the surface plate to move the dial indicator across the workpiece surface. Full indicator movement (FIM) of less than or equal to .005" shows that the perpendicularity tolerance has been met. The workpiece must be clamped in a fixed location while the surface measurement is being made.

If the workpiece fails the first attempt to check the perpendicularity tolerance, it does not mean the part is bad. Because only one datum is referenced in the feature control frame, the workpiece may be rotated on the angle block and clamped in place for another measurement.

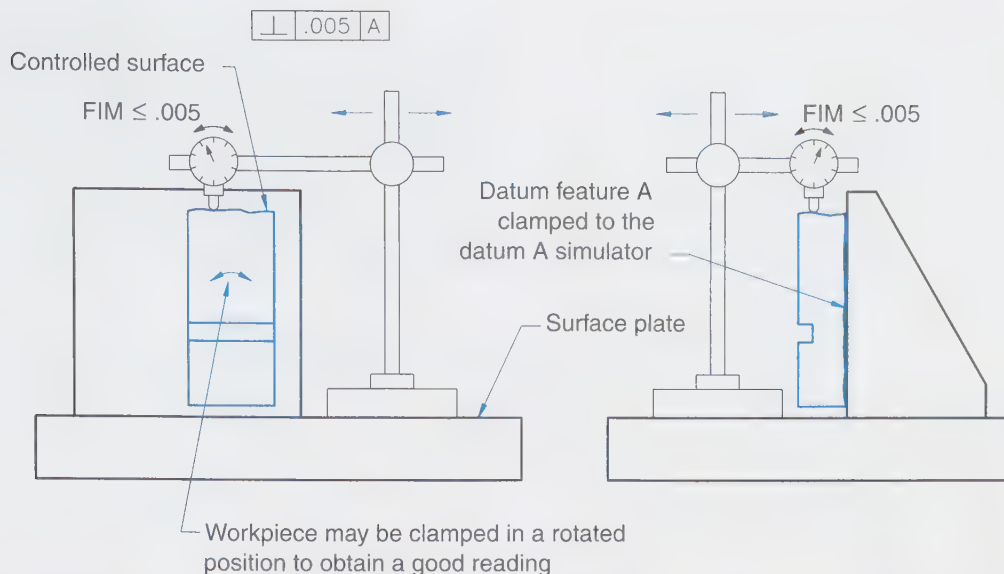
The workpiece may be rotated on the angle block because the perpendicularity tolerance is only controlled relative to datum A. The proper relationship relative to datum A is maintained if the workpiece is clamped to the angle block. Perpendicularity to datum A is not affected by rotation of the workpiece on datum A. If the full indicator movement cannot be minimized to .005" or less, the part is bad. If the readings are brought to within .005", the perpendicularity requirement has been met.

There are additional requirements on the surface that may limit the angle of the surface relative to other sides of the part. There may be a general angularity tolerance shown in the title block or notes. The general angularity tolerance will establish the allowable angle variation of the end surface relative to the sides of the part. However, this



Goodheart-Willcox Publisher

Figure 7-14. The tolerance zone for a perpendicularity tolerance on a flat surface is bounded by two planes.



Goodheart-Willcox Publisher

Figure 7-15. The workpiece may be rotated on the datum A simulator to optimize the perpendicularity measurement.

may not be the best method for establishing additional requirements beyond the perpendicularity tolerance that is already shown.

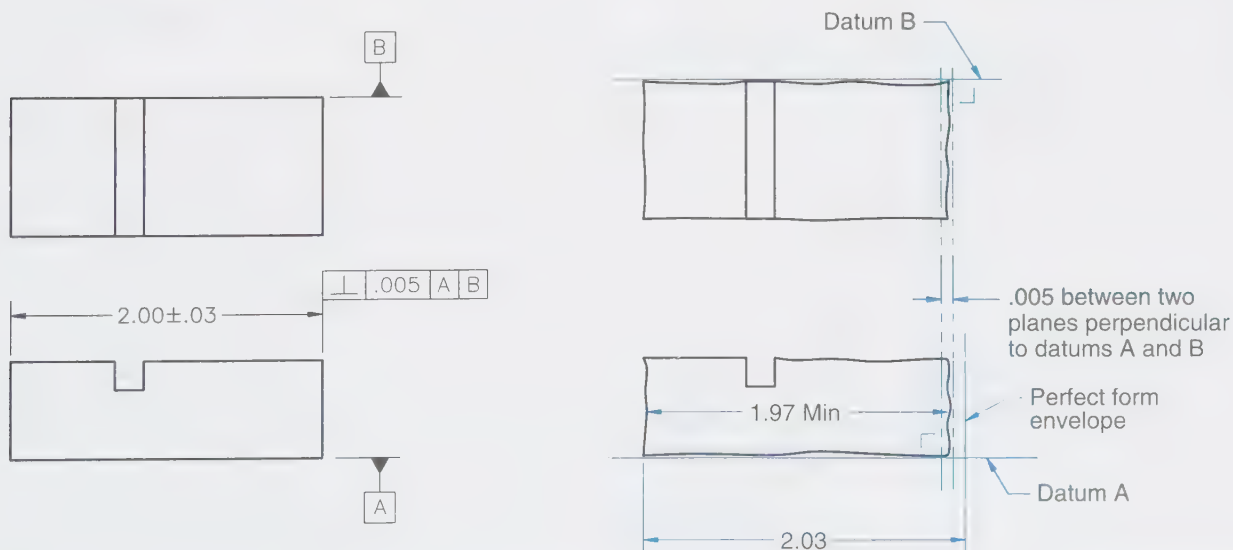
If it is desirable to constrain the degrees of freedom for the perpendicularity tolerances relative to one of the sides of the part, a secondary datum feature reference must be specified in the feature control frame. This is explained in the following section.

It is important to note that the perpendicularity tolerance also requires that the surface form be contained within the tolerance zone. Any surface that lies within an orientation tolerance zone also has a form that is equal to or better than the orientation

tolerance. If the surface in the given example lies between two orientation tolerance zone planes that are separated by .005", flatness has also been controlled to within .005". It is not necessary to apply a form tolerance to this surface unless a form tolerance less than .005" is needed.

Referenced to Two Datum Features

A perpendicularity tolerance may be referenced to two datum features. See **Figure 7-16**. Referencing two datum features, when the datum features are flat surfaces, constrains all rotational degrees of freedom. The tolerance zone cannot rotate, but it is free to translate (move). Referencing two datums is a more stringent control than



Goodheart-Willcox Publisher

Figure 7-16. A perpendicularity tolerance may be referenced to two datums.

referencing one datum, but it is typically appropriate for parts of the type shown in the figure. Referencing two datums for an orientation tolerance reduces reliance on directly applied angle tolerances.

It is necessary to identify the two datum features on a part when two datum features are referenced in a feature control frame. The given figure has datum features A and B identified. The feature control frame for the perpendicularity tolerance references datum feature A as primary and datum feature B as secondary. For a perpendicularity tolerance, the primary datum feature must be at a 90° angle to the toleranced feature. The secondary datum may be established by any feature that prevents (constrains) rotation of the part while it is located on the primary datum.

The perpendicularity tolerance zone for the toleranced surface on the given part is bounded by two planes separated by .005". The planes are perpendicular to both datum planes A and B.

The tolerance zone may be verified by clamping datum surface A of the workpiece against an angle block (datum A simulator) that is set on a surface plate. See Figure 7-17. The workpiece must be clamped against the datum A simulator with datum feature B making two point contact with a second angle block (datum B simulator). Clamping the workpiece against the two angle blocks establishes the required orientation of the part. The workpiece must remain in this position while the perpendicularity measurements are made.

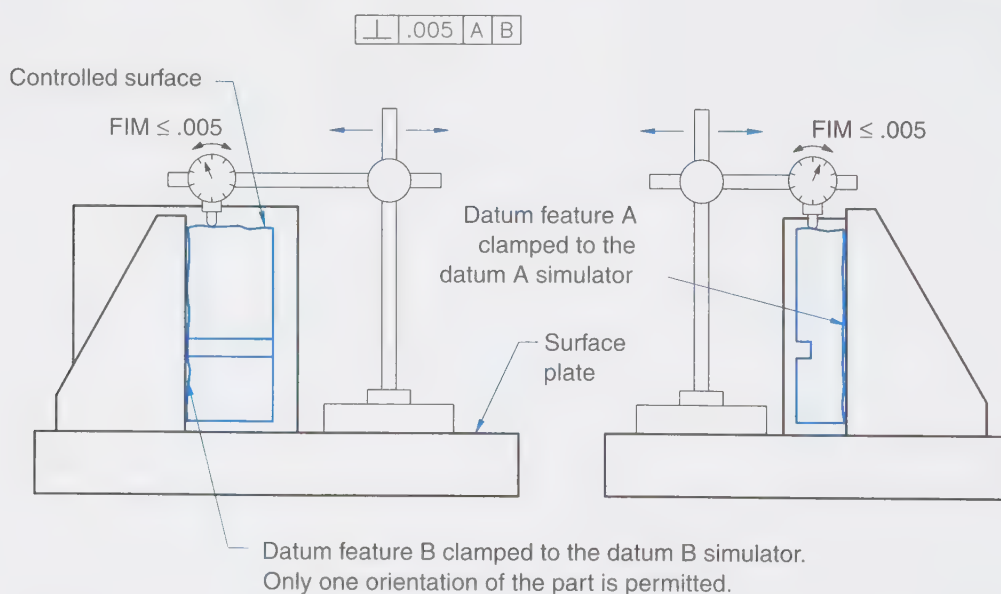
A dial indicator mounted in a height stand can be moved across the controlled surface. A full indicator movement of .005" or less indicates the perpendicularity tolerance has been met. It is not acceptable to rotate the part in an attempt to improve the indicator reading because primary and secondary datums are referenced. A reading of more than .005" indicates the feature is out of tolerance.

Face Surface Elements on a Shaft

Perpendicularity of the face surface on a shaft may be tolerated relative to the shaft axis. The feature control frame in an orthographic view is attached to an extension line or leader from the surface. See Figure 7-18. In a solid model, the feature control frame is connected by a leader that terminates with a dot placed on the surface.

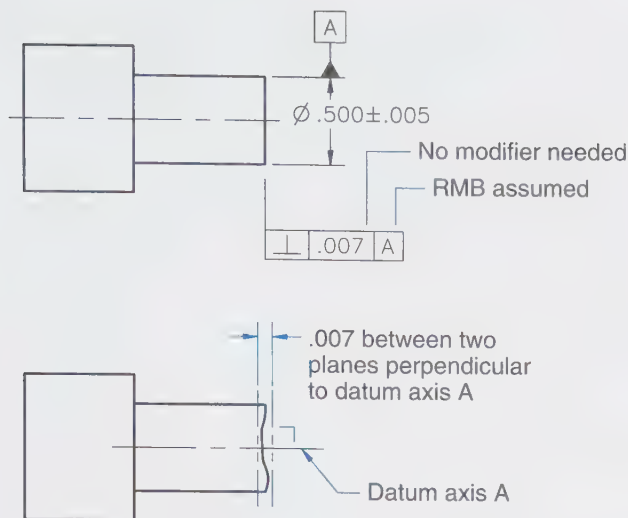
The shown feature control frame references datum feature A. The shaft diameter is identified as datum feature A. Datum feature A is a feature of size. The datum feature reference in the feature control frame does not show a material boundary modifier. Therefore, it is assumed to apply regardless of material boundary.

Two planes establish the tolerance zone for the given part. They are separated by .007" and are perpendicular to the datum axis. The surface may lie anywhere between the two planes. A similar tolerance requirement may be achieved using profile and runout tolerances.



Goodheart-Willcox Publisher

Figure 7-17. All referenced datum features must be properly simulated when checking a tolerance specification.



Goodheart-Willcox Publisher

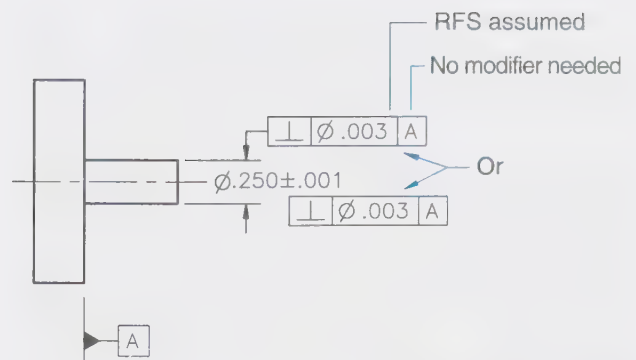
Figure 7-18. Perpendicularity of a flat surface relative to a datum axis may be specified.

Note

The ASME Y14.5-2009 standard permits the control of perpendicularity of individual radial line elements relative to a datum axis. This practice using perpendicularity is not recommended because it also implies a location of the radial line relative to the datum axis. (It can only be a radial line if it is located to pass through the axis.) Rather than use an orientation tolerance for this type of tolerance requirement, line profile as explained in a later chapter can achieve greater clarity of the tolerance requirement.

Perpendicularity Applied to Features of Size

Perpendicularity applied to a feature of size establishes control of the center plane or axis of the feature. The feature control frame is associated with the feature of size. This may be done in orthographic views or solid models by placing the feature control frame adjacent to the size dimension value, or it may be attached to the dimension line. See Figure 7-19. In a solid model, the feature control frame may be connected by a leader that terminates with a dot on the surface of a cylinder. The presence of the diameter symbol indicates that the tolerance is controlling the axis and that the zone is cylindrical.



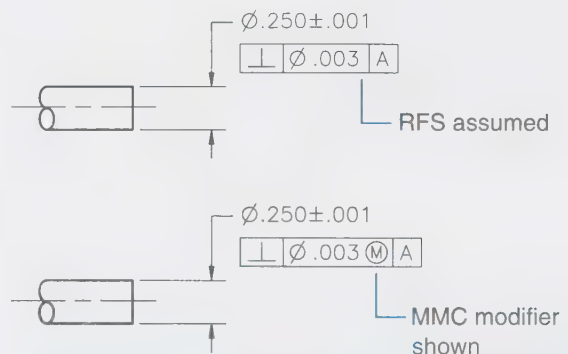
Goodheart-Willcox Publisher

Figure 7-19. An axis of a feature may be specified to be perpendicular to a plane surface.

Material Condition Modifiers and Virtual Condition

A material condition modifier applies to each orientation tolerance that is applied to a feature of size. See Figure 7-20. If no material condition modifier is shown following the tolerance value, then RFS is assumed. Either the MMC or LMC modifier must be shown if they are to apply.

When applied RFS on a cylinder, a cylindrical tolerance of the specified diameter is created and the axis must lie within the zone. When applied MMC, the cylindrical tolerance zone is created and has a size equal to the specified tolerance when the feature is at MMC. A virtual condition is also created by the MMC application and the surface of the feature must not violate the virtual condition boundary. LMC creates a tolerance zone that is applicable when the feature is at the LMC size and it also has a virtual condition that must not be violated.



Goodheart-Willcox Publisher

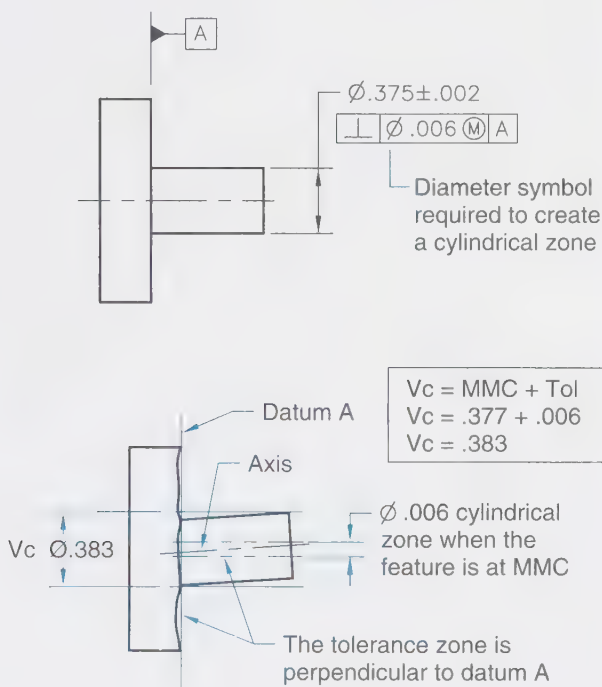
Figure 7-20. Orientation tolerances applied to features of size are assumed to apply RFS unless specified otherwise.

Controlling Axis Orientation of a Pin

The perpendicularity of a pin may be tolerated relative to a datum feature. See **Figure 7-21**. This figure shows a pin tolerated relative to a flat surface that is identified as datum feature A. The perpendicularity feature control frame is placed adjacent to the .375" diameter dimension.

A diameter symbol is shown in front of the tolerance value to indicate the tolerance value is the diameter of a cylindrical zone. The diameter symbol is generally required when an orientation tolerance applies to a cylindrical feature of size. (An exception would be when controlling the perpendicularity in only one plane.)

In the given figure, the amount of perpendicularity tolerance shown in the feature control frame is .006" when the feature is at MMC. The lower half of the figure shows a .006" diameter cylindrical tolerance zone, when the pin is at its MMC size, extending perpendicular to datum A. This cylinder defines the boundary inside of



	Pin size	Allowable perp. tolerance
MMC	.377	.006
	.376	.007
	.375	.008
	.374	.009
LMC	.373	.010

Goodheart-Willcox Publisher

Figure 7-21. The virtual condition for an external feature of size is larger than the MMC of the feature.

which the axis of the pin must fall when the pin is at its MMC. If the pin departs from MMC, then the tolerance increases, allowing more perpendicularity variation. The axis of the pin, as defined by the ASME standard, is the axis of the *unrelated* actual mating envelope. Because it is the axis of the unrelated mating envelope that is controlled, the perpendicularity tolerance has no direct effect on the straightness or orientation of the derived median line of the pin. Derived median line straightness was explained in Chapter 5.

Because the tolerance includes an MMC modifier, the cylindrical tolerance zone has a diameter of .006" when the pin is at MMC (.377"). As the actual mating envelope of the pin departs in size from the .377" MMC, the tolerance zone inside of which the axis must fall is permitted to increase. For every .001" departure from MMC, there is an equal amount of increase in the tolerance zone.

Checking axis orientation is often adequate, but when substantial form variation exists on the tolerated feature, the axis method may reject some acceptable parts. Checking the surface of the pin relative to a virtual condition boundary will allow acceptance of all functional parts. The surface method is explained in the following paragraphs.

The combined effect of the MMC size of the pin and the orientation tolerance creates a virtual condition boundary as shown in **Figure 7-21**. The surface of the pin must not violate the virtual condition boundary. This is referred to as the surface (boundary) method and takes precedence over the axis method. In situations where substantial form variation exists, it is sometimes possible for the axis method to give different results than the surface method. Should there be a significant difference between the two, the surface method takes precedence. This is explained in greater detail in the following paragraphs.

Using a feature control frame is the clearest way to specify a perpendicularity tolerance on an individual feature. If no feature control frame is shown, then another means of tolerance specification is required to establish some level of orientation control on the 90° angles. It may be through general notes or title block tolerances. A perpendicularity tolerance or another form of tolerance is required because size tolerances, such as for a pin diameter, only define allowable size and form variation; they do not define allowable orientation tolerance.

Virtual Condition of a Pin

Orientation tolerances with the MMC modifier result in a virtual condition. When a perpendicularity tolerance is applied to a pin, the virtual

condition is equal to the apparent size of the pin when it is at MMC and the maximum permitted orientation variation exists.

Virtual condition may be thought of as the required size for a mating part, assuming the mating part is perfect. In the case of a pin that includes an orientation tolerance, the virtual condition is the size of a perfect hole that fits over the pin when the pin is at MMC and is in the worst permitted orientation.

Figure 7-22 shows the virtual condition for the pin in the previous figure. The virtual condition is determined by adding the MMC size and the orientation tolerance. A pin with a .377" MMC size and a perpendicularity tolerance of .006" has a virtual condition of .383" diameter.

A theoretical gage can be produced at the virtual condition size to check the orientation of the pin. Any pin that has an acceptable size and an acceptable orientation will fit into the gage with datum feature A in contact with at least three points. Acceptable orientations are those that result

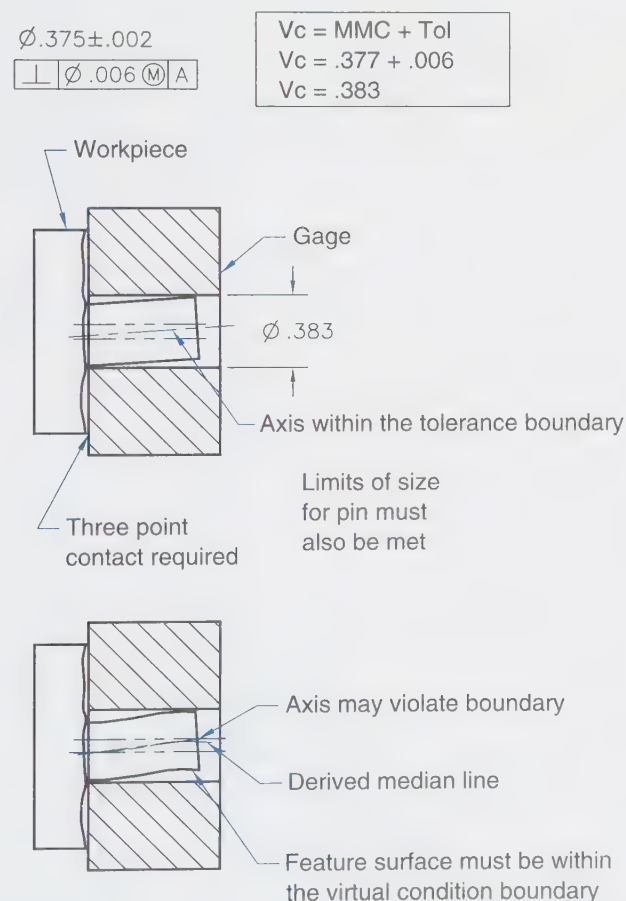
in the feature surface not violating the cylindrical virtual condition boundary. There is no requirement of location created by the perpendicularity tolerance.

It was previously explained that a tolerance with the MMC modifier is permitted to increase as the tolerated feature departs from MMC. When using the surface (boundary) method of verification, the orientation tolerance of .006" at MMC increases as the pin departs from MMC. When the pin is at a diameter of .376", there is a .001" departure from MMC. This permits the orientation tolerance to increase by .001" to a total permitted tolerance of .007". See the table in Figure 7-21.

A produced pin diameter of .376" and orientation tolerance of .007" combine to cause a *related actual mating envelope* diameter of .383" diameter, and that is consistent with the virtual condition boundary that the pin must remain within. The mating envelope is considered "related" because it is constrained in rotation (related) to the referenced datum(s). When the related actual mating envelope of a pin is equal to or smaller than the virtual condition, the pin orientation is good.

Referring to Figure 7-22, only one fixed diameter gage with a size equal to the virtual condition is needed to inspect the orientation tolerance on any pin that is made within the specified tolerances. The shown gage only checks perpendicularity. It does not check the size of the pin.

The bottom half of the given figure shows a pin that has departed from the MMC size. Size departure from the boundary of perfect form at MMC permits form variation on the pin as well as additional orientation tolerance. The pin with straightness variation has both an axis and a derived median line. The axis is the center of the unrelated actual mating envelope for the pin, and the axis is required to be within the perpendicularity tolerance. The derived median line is formed along the center of each cross section along the pin and is not straight if the pin is not straight. Perpendicularity tolerances are not applicable to the derived median line.



Goodheart-Willcox Publisher

Figure 7-22. A functional gage for an external feature is sized on the basis of the virtual condition of the feature.

Note

A fixed diameter gage may only be used when the tolerance includes the MMC modifier.

For a clearance fit, a mating hole must have tolerances that result in a virtual condition diameter no smaller than the virtual condition of the pin. This ensures that 100% of the parts produced

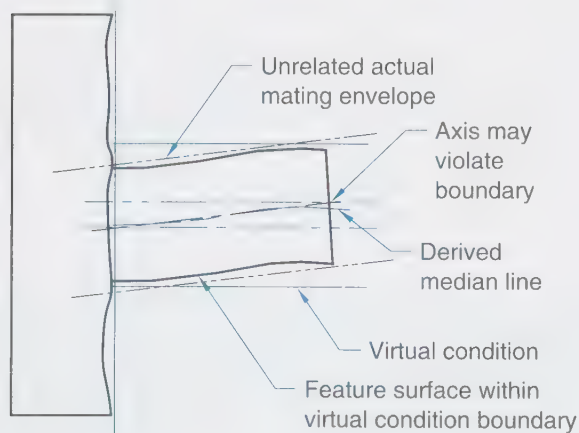
within the tolerances will assemble when a worst-case condition exists.

The preceding paragraphs explained two methods for interpretation of orientation tolerances. The first (the axis method) explained the cylindrical tolerance zone that is to contain the axis of the tolerated feature. The second (the surface method) described how the size and orientation tolerances combine to create a virtual condition that the feature must fit within. When the form of the feature is perfect, the requirements for the axis and for the surface are identical in terms of the allowable variation. However, if the tolerated feature departs from its true geometric shape, the axis and surface methods may give different results. Should this take place, the surface method takes precedence.

The bottom half of **Figure 7-22** shows a pin that is not straight. This same illustration is included at an increased scale in **Figure 7-23**. Because of form variation in the shown pin, the axis of the pin (the axis of the unrelated actual mating envelope) may go slightly outside the tolerance zone. However, the surface of the pin does not violate the virtual condition boundary. The pin has therefore met the perpendicularity tolerance requirement.

Controlling Axis Orientation of a Hole

The perpendicularity of a hole may be controlled relative to a datum feature. See **Figure 7-24**. This figure shows an orientation tolerance for a hole tolerated relative to a flat surface that is identified as datum feature A. The feature control frame is placed adjacent to the .392" diameter dimension.



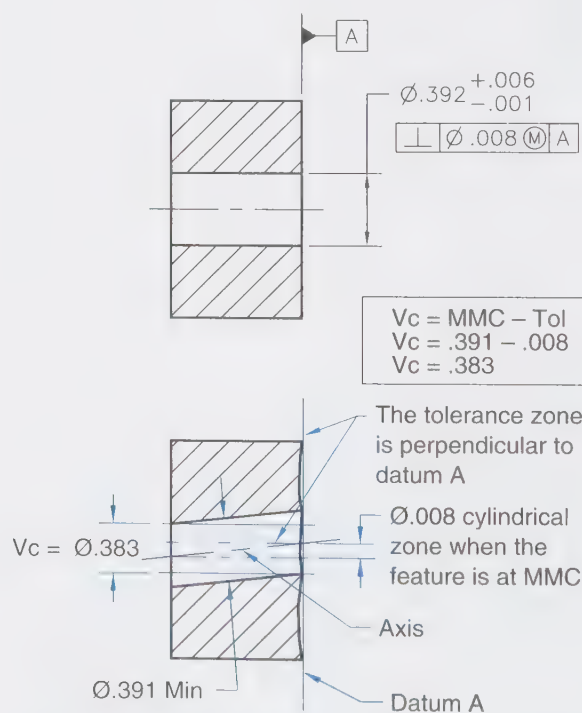
Goodheart-Willcox Publisher

Figure 7-23. The axis of the tolerated feature is at the center of the unrelated actual mating envelope and is straight. The derived median line is at the center of the feature and may not be straight when form variation exists.

A diameter symbol is shown in front of the tolerance value to indicate the tolerance value is the diameter of a cylindrical zone. The diameter symbol is generally required when an orientation tolerance applies to a cylindrical feature of size.

In the given figure, the amount of perpendicularity tolerance shown in the feature control frame is .008" when the feature is at MMC. The lower half of the figure shows a .008" diameter cylindrical tolerance zone, when the hole is at its MMC, extending perpendicular to datum A. This cylinder defines the boundary inside of which the axis of the hole must fall. If the hole departs from its .391" MMC, then the tolerance increases, allowing more perpendicularity variation. The axis of the hole, as defined by the ASME standard, is the axis of the *unrelated* actual mating envelope. Because it is the axis of the unrelated mating envelope that is controlled, the perpendicularity tolerance has no direct effect on the derived median line straightness within the hole.

Because the tolerance includes an MMC modifier, the cylindrical tolerance zone has a diameter of .008" when the hole is at MMC (.391"). As the hole departs in size from the .391" MMC, the tolerance zone is permitted to increase. For every .001" departure from MMC, there is an equal amount of increase in the tolerance zone.



Goodheart-Willcox Publisher

Figure 7-24. The virtual condition for an internal feature of size is smaller than the MMC of the feature.

The combined effect of the MMC size of the hole and the orientation tolerance creates a virtual condition boundary as shown in **Figure 7-24**. The surface of the hole must not violate the virtual condition boundary.

Using a feature control frame is the clearest way to specify a perpendicularity tolerance on an individual feature. If no feature control frame is shown, then another means of tolerance specification is required to establish some level of orientation control on the 90° angles. It may be through general notes or title block tolerances. A perpendicularity tolerance or another form of tolerance is required because size tolerances, such as for a hole diameter, only define allowable size and form variation; they do not define allowable orientation tolerance.

Virtual Condition of a Hole

Orientation tolerances that include an MMC modifier result in a virtual condition. When a perpendicularity tolerance is applied to a hole, the virtual condition is equal to the apparent size of the hole when it is at MMC and the maximum permitted orientation variation exists.

The virtual condition of a hole is the size of a perfect pin that would fit into the leaning hole. A mating pin must have a virtual diameter equal to the virtual condition of the hole to ensure the parts will assemble when the worst-case condition exists.

Figure 7-25 shows the virtual condition for the part in the previous figure. The virtual condition is determined by subtracting the specified orientation tolerance from the MMC size. A hole with

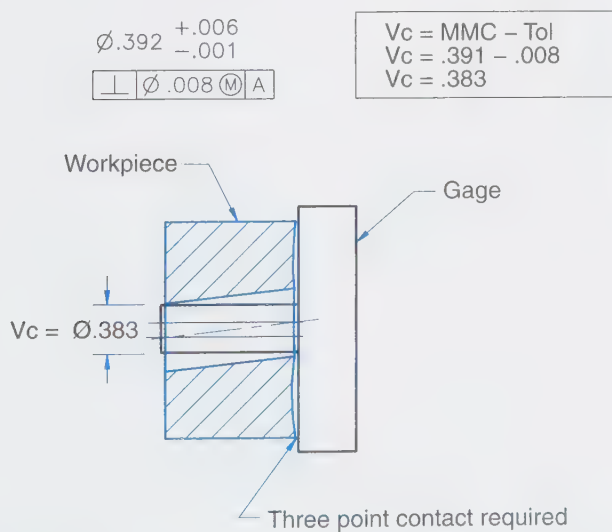
a .391" MMC size and a perpendicularity tolerance of .008" has a virtual condition of .383" diameter.

A theoretical gage may be produced at the virtual condition size to check the orientation of the hole. Any hole that has an acceptable size and an acceptable orientation will fit onto the gage with datum feature A in contact with at least three points. Acceptable orientations are those that result in the feature surface not violating the cylindrical virtual condition boundary. There is no requirement of location created by the perpendicularity tolerance.

It was previously explained that a tolerance with the MMC modifier is permitted to increase as the toleranced feature departs from MMC. On the given part, the orientation tolerance of .008" at MMC increases if the hole departs from MMC. When the hole is at a diameter of .392", there is a .001" departure from MMC. This permits the orientation tolerance to increase by .001" to a total permitted tolerance of .009".

A produced hole diameter of .392" and orientation tolerance of .009" result in a related actual mating envelope diameter of .383" diameter. This is equal to the virtual condition boundary that the hole must not violate. When the related actual mating envelope of a hole is equal to or larger than the virtual condition, the orientation of the hole is good.

One fixed diameter gage with a size equal to the virtual condition is needed to inspect the orientation tolerance on any hole that is made within the specified size tolerance. The shown gage only checks perpendicularity. It does not check the size or location of the hole.



Goodheart-Willcox Publisher

Figure 7-25. A functional gage for a hole is sized on the basis of the virtual condition of the hole.

Note

A fixed diameter gage may only be used if the tolerance includes the MMC modifier.

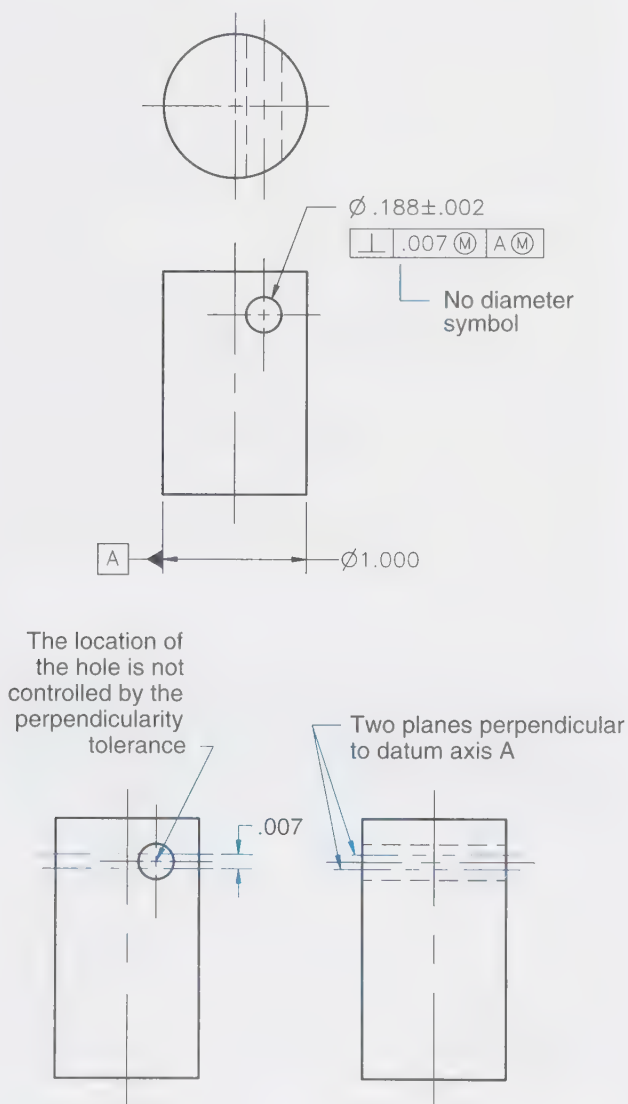
As explained in previous chapters, a feature may have form variation when the feature has departed from MMC. If a hole has form variations that are within the limits of size and that hole also has orientation variations, the combined variations are acceptable if the surface of the hole does not violate the virtual condition. Checking the same hole using the axis method could potentially show the hole to be unacceptable. In such situations, the hole is acceptable. The surface requirement and surface method of verification takes precedence over the axis method.

Controlling Two Perpendicular Cylinders

There are exceptions to the requirement for a diameter symbol on perpendicularity tolerances applied to cylinders. See [Figure 7-26](#). No diameter symbol is required on a perpendicularity tolerance when applied to a cylindrical feature and referenced to a datum axis.

The shaft diameter in the given figure is identified as datum feature A. The hole has a perpendicularity tolerance of .007" referenced to datum feature A. There is no diameter symbol on the tolerance value. The tolerance value has an MMC modifier and the datum feature reference has the MMB modifier.

The tolerance zone is bounded by two planes that are perpendicular to datum axis A. The planes



Goodheart-Willcox Publisher

Figure 7-26. When a perpendicularity tolerance without a diameter symbol is applied to control the orientation of a feature axis relative to a datum axis, the tolerance zone is bounded by two parallel planes.

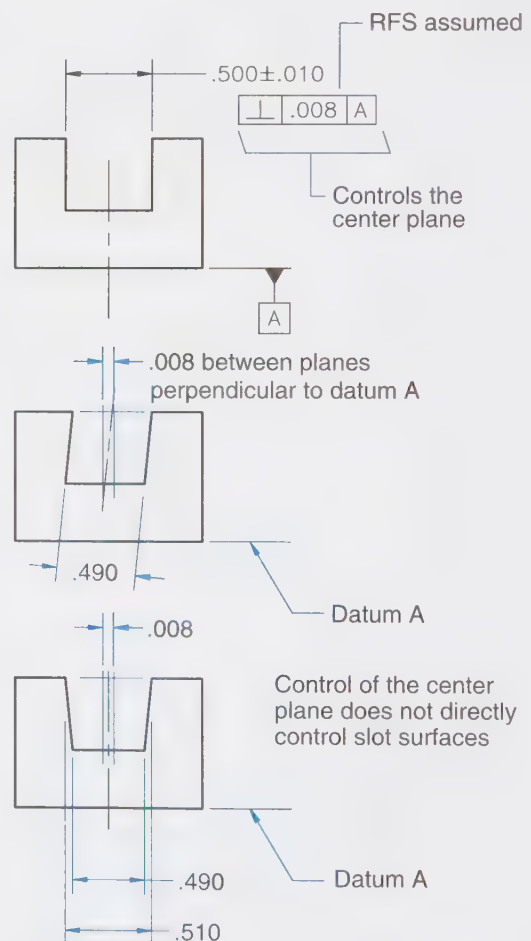
are separated by .007" when the hole is at the MMC size. If the axis of the hole is anywhere within the two planes, it meets the perpendicularity requirement relative to the datum axis.

The orientation tolerance does not control the location of the hole relative to the datum axis. It only controls the orientation. Location tolerance for the hole would be specified using a position tolerance.

Controlling Perpendicularity of a Center Plane

It is sometimes necessary to specify the allowable perpendicularity tolerance for the center plane of a rectangular feature of size. See [Figure 7-27](#). In the given figure, the perpendicularity of the center plane of a slot is toleranced. The same method of tolerancing the perpendicularity of a center plane may be applied to external features of size such as rails and tabs.

The feature control frame is placed adjacent to the size dimension for the slot. This indicates



Goodheart-Willcox Publisher

Figure 7-27. An orientation tolerance applied to a rectangular feature of size controls the orientation of the center plane of the feature.

that the feature of size is controlled, which in effect controls the center plane. The specified tolerance is .008" RFS relative to datum A. Datum feature A is the bottom surface of the part.

Only the center plane of the slot is controlled by the given perpendicularity tolerance. The bottom of the slot is not controlled by this tolerance. Because the tolerance in this example is applied RFS, the tolerance applies to the center plane, and only the center plane method may be used to verify the tolerance is met. There is no virtual condition created, so the surface method cannot be used. If the tolerance were applied MMC, a virtual condition would exist and the surface method would be applicable.

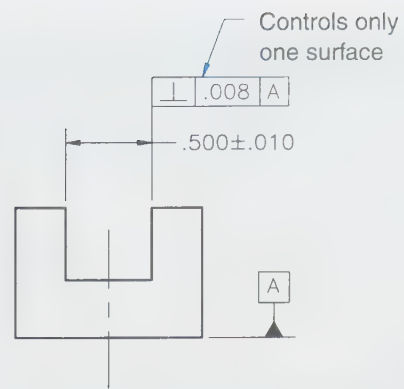
The sides of the slot may have form variation or angular variation (be in any condition) provided that Rule #1 and the .490" and .510" limits of size are met. Rule #1 requires the sides of the slot be parallel to one another and flat when produced at the MMC size of .490". However, the size limits do not require the sides of the slot to be perpendicular to datum A. On the given drawing, only the feature control frame invokes a perpendicularity tolerance requirement.

Regardless of the produced size of the slot, the center plane must be within a boundary established by two planes perpendicular to datum A and separated .008". Any orientation of the center plane within the boundary is acceptable. The tolerance zone extends the full height of the slot.

Two possible and acceptable conditions for the slot are shown in the given figure. The first example shows a slot produced at the MMC size of .490". The two sides of the slot are parallel, but oriented such that the center plane stays just inside the tolerance boundary. Because the slot is at MMC, the perpendicularity tolerance indirectly affects the sides of the slot.

The second example shows the slot produced with a .490" width at the bottom and a .510" width at the top. In this example, the perpendicularity of the center plane is perfect, but the sides of the slot are at considerable angles to datum A. This part meets both the size and center plane perpendicularity requirements.

Control of the orientation of a center plane at RFS in combination with the size tolerances does have an impact on the two surfaces of the slot. However, the impact is difficult to assess. To directly tolerance the perpendicularity of the sides of a slot, a feature control frame may be connected to each slot surface that is to be tolerated. See **Figure 7-28**. In orthographic views, the feature



Goodheart-Willcox Publisher

Figure 7-28. A feature control frame placed on an extension line controls only the surface from which the line extends.

control frame is attached to an extension line from the surface to be controlled. In orthographic views and solid models, it is acceptable to use a leader to attach a feature control frame to a surface.

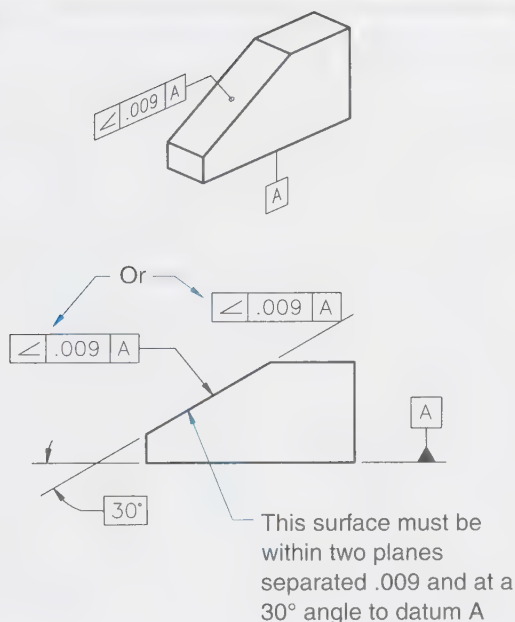
The shown feature control frame in **Figure 7-28** is applied to only one surface. It only controls the surface to which it is applied. The other side of the slot is controlled by the limits of size on the slot. If direct control of both surfaces is desired, a feature control frame may be applied to each of them.

Another means of controlling the sides of a slot, or other feature of size, is to specify an orientation tolerance at MMC. The orientation tolerance at MMC establishes a virtual condition that the sides of the slot must not violate. The orientation tolerance in combination with the size limits establishes the amount of variation that may be present.

Angularity

Control of any flat surface at an angle other than parallel or perpendicular to the referenced datums may be established with an angularity tolerance. Angularity tolerances are always referenced to one or more datums. See **Figure 7-29**. In an orthographic view, the feature control frame is attached to a leader or extension line from the surface to be controlled. In solid models, the feature control frame is attached to the surface with a leader that terminates at a dot on the surface.

The angle dimension(s) that defines the surface orientation must be basic. No angles are assumed other than perpendicularity and parallelism, as already explained. Angles aren't always dimensioned in solid models, but the queried angle between the datum feature and the tolerated feature establishes the basic angle.



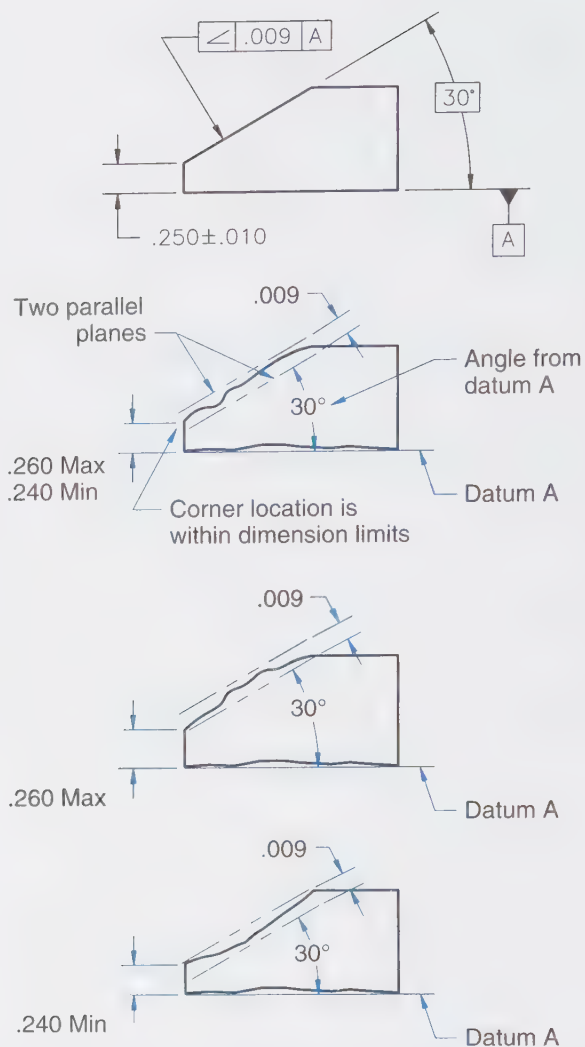
Goodheart-Willcox Publisher

Figure 7-29. A feature control frame for an angularity tolerance may be placed on an extension line or leader extending from the surface to be controlled.

Angularity Applied to Flat Surfaces

Application and interpretation of an angularity tolerance is shown in **Figure 7-30**. The vertex of the angle is located with a toleranced dimension. The angle of 30° is shown as basic. Datum feature A is identified. An angularity tolerance of $.009''$ relative to datum A is specified for the inclined surface. The tolerance zone for the given figure is bounded by two parallel planes separated $.009''$. These planes are at an angle of 30° relative to datum A.

Surface conditions (form and orientation) are only controlled by the angularity tolerance to the extent that all points on the surface must fall within the $.009''$ wide zone. This tolerance zone is free to



Goodheart-Willcox Publisher

Figure 7-30. The angle dimension defining the surface orientation must be basic when an angularity tolerance is applied.

float under the condition that it remain at a 30° angle to datum A, and the corner on the part must fall somewhere within the limits defined by the dimension applied. The corner of the given part may fall anywhere within the limits of $.240''$ and $.260''$.

Some controversy exists regarding the allowable location of the planes that define the orientation tolerance zone. To avoid controversy regarding the effect of the plus or minus tolerance defining the location of the surface, a basic dimension may be used to locate the corner and a profile tolerance applied as defined in a later chapter.

The following explanation is based on the premise that orientation tolerances control only orientation when applied to a surface; they do not control location. Should a need exist to simultaneously control location and orientation requirements, a profile tolerance may be specified.

It is possible for the corner of the part shown in **Figure 7-30** to be located at .260" and have the bottom plane of the angularity tolerance zone pass through. It is also possible to have the corner at .240" with the top plane of the angularity tolerance zone passing through. Anything between these two extremes is also permitted.

Any surface that lies within the orientation tolerance zone also meets a form tolerance that is equal to the orientation tolerance. The given figure shows this effect. The tolerance zone is defined by two parallel planes. If the surface is between the planes, then form is controlled.

Referenced to One Datum Feature

An angularity tolerance referenced to one datum only constrains the rotational degrees of freedom for the orientation tolerance zone relative to that datum. See **Figure 7-31**. The given angularity tolerance specifies an orientation requirement of .009" relative to datum A. The angularity tolerance does not control the surface relative to any feature other than datum feature A, nor does it control the location of the surface. Location requirements are defined by a directly toleranced dimension or a profile tolerance. Profile tolerances are the preferred method of defining location tolerances on surfaces and are defined in a later chapter.

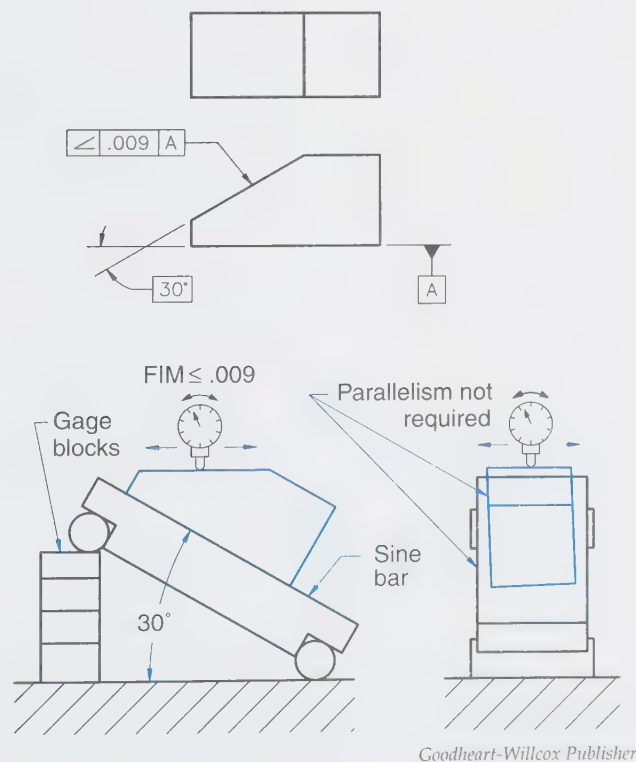


Figure 7-31. Angularity must be verified relative to the datums that are referenced. No datums are to be assumed.

The given angularity tolerance specification does not constrain rotation of the controlled surface around any axis that is perpendicular to datum A. Rotation around such an axis does not affect the 30° angle relative to datum A.

The shown method of verification for the given angularity tolerance illustrates the constrained and unconstrained rotations of the surface. A sine bar is set on gage blocks to orient it at a 30° angle. The workpiece is placed on the sine bar in a position that results in the specified surface being horizontal. This constrains two degrees of rotation of the part relative to datum plane A. It remains free to rotate on an axis perpendicular to datum plane A.

A dial indicator is passed across the workpiece to determine if the orientation requirement is met. A full indicator movement of .009" or less indicates the part is good.

The workpiece in the given figure is shown such that it is not aligned with the edge of the sine bar. This is an acceptable position because only one datum is referenced. Control of the angularity tolerance zone is specified relative to only datum A, and resting the bottom surface of the part on the sine bar establishes orientation relative to datum A.

Although the feature control frame does not establish orientation requirements other than to datum A, there is some limit to the permitted rotation. A complete drawing of the given part would include size dimensions for all features and general angularity tolerances. These size and angularity tolerances would provide a limit to the amount of rotation of the subject surface. However, the angularity tolerance itself is free to rotate to confirm that the .009" tolerance is met relative to datum A. Other requirements that may affect the surface are checked separately.

The degree of orientation control established by size dimensions on angled surfaces is difficult to calculate and often ambiguous. If the orientation tolerance zone needs to be well defined in relationship to more than the primary datum, an angularity tolerance should be specified that references two datum features.

Referenced to Two Datum Features

Figure 7-32 shows the same part as the previous figure except the feature control frame references two datum features and a second datum feature is identified. The feature control frame shows datum A as primary and datum B as secondary. Datum feature A is the bottom of the part and is the surface from which the angle is dimensioned. Datum feature B is one side of the part.

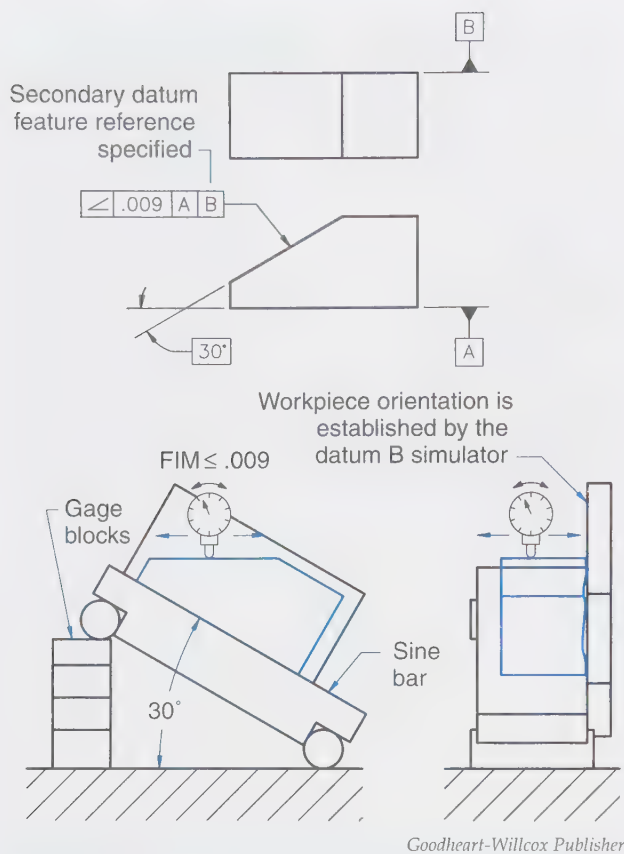


Figure 7-32. All referenced datums must be simulated when checking angularity tolerances.

The workpiece is set on the same sine bar that was shown in the previous figure. As shown previously, the primary datum A constrains two degrees of freedom. The secondary datum feature reference constrains the third rotational degree of freedom. The difference in verification methods is that datum feature B must be properly oriented on the sine bar. For this part, datum feature B is aligned with the edge of the sine bar. With the workpiece properly oriented on the sine bar, a dial indicator is moved across the surface. A full indicator movement of .009" or less indicates a good part.

The orientation of the part may not be changed to reduce the indicator movement as it may when only one datum feature is referenced. If a reading greater than .009" is obtained, the surface does not meet the specified tolerance and the part should be rejected or reworked.

Tangent Plane

All orientation tolerances, when applied to a surface, establish a tolerance boundary inside of which the controlled surface must lie. Any form or orientation variation that is contained within the boundary is permitted.

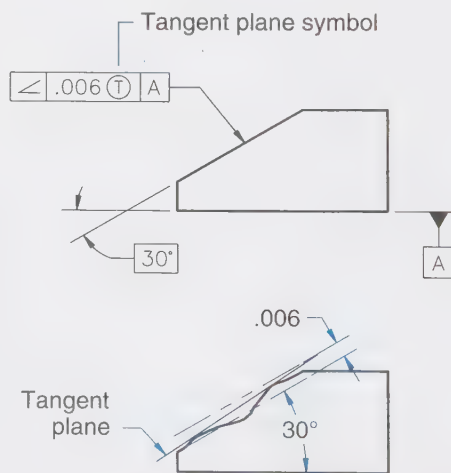
One extreme possible surface condition permitted by a .010" orientation tolerance is an abrupt .010" step at one location on the surface. Another permitted condition is for the surface to be flat and at an orientation that extends across the tolerance zone.

Significantly different effects are caused when a mating part is located against the two surface conditions described in the previous paragraph. The angle of the mating part is affected by the amount and location of the surface variations.

It is possible to control the effects of surface conditions when applying an angularity tolerance. One method for reducing the orientation effects caused by surface variation is to apply a form tolerance in addition to the orientation tolerance. Another method is to specify the orientation tolerance in a way that makes it applicable to a tangent plane. See [Figure 7-33](#). This is done by placing the tangent plane symbol in the tolerance specification following the tolerance value. This may be done for parallelism, perpendicularity, and angularity tolerances.

Specifying tolerances for a tangent plane should only be done when it is important to control the tangent plane. An orientation tolerance on a tangent plane may in some instances be more difficult to verify than an orientation tolerance on a surface. If an orientation tolerance applied to the part surface results in adequate control, the tangent plane requirement should not be used.

The angularity tolerance zone for a tangent plane is identical to any other angularity tolerance zone, except the surface of the controlled feature is not required to entirely fall within the tolerance zone boundary. A plane that is tangent to the high



Goodheart-Willcox Publisher

Figure 7-33. Angularity may be specified to control the plane that is tangent to a surface.

points on the surface must be within the tolerance zone. The surface itself is not controlled except at the high points that contact the tangent plane.

The given figure shows a tolerance zone that contains a tangent plane. The surface itself extends outside the tolerance zone. Orientation tolerances specified for a tangent plane do not control the surface form. In fact, the tolerance requirements for the form of the surface are completely undefined by the tangent plane tolerance. This means that a separate form or profile tolerance must be applied to the surface if form control is desired. To omit a form tolerance from the surface would in most applications be an error. The following paragraphs explain how form tolerances may be shown along with orientation tolerances.

Combined Form and Orientation Tolerances

Multiple feature control frames may be applied to a single feature. This is often done to control orientation to one value, and form to a smaller value. Combined orientation and form tolerances are only required on surfaces when the maximum allowable form tolerance is less than the allowable orientation tolerance or when the orientation is applicable to a tangent plane. An orientation tolerance applied to a surface requires the surface form to be as good as the orientation tolerance. Therefore, there is no need to apply a form tolerance unless the form requirement is smaller than the orientation tolerance.

Orientation Also Controls Form on Surfaces

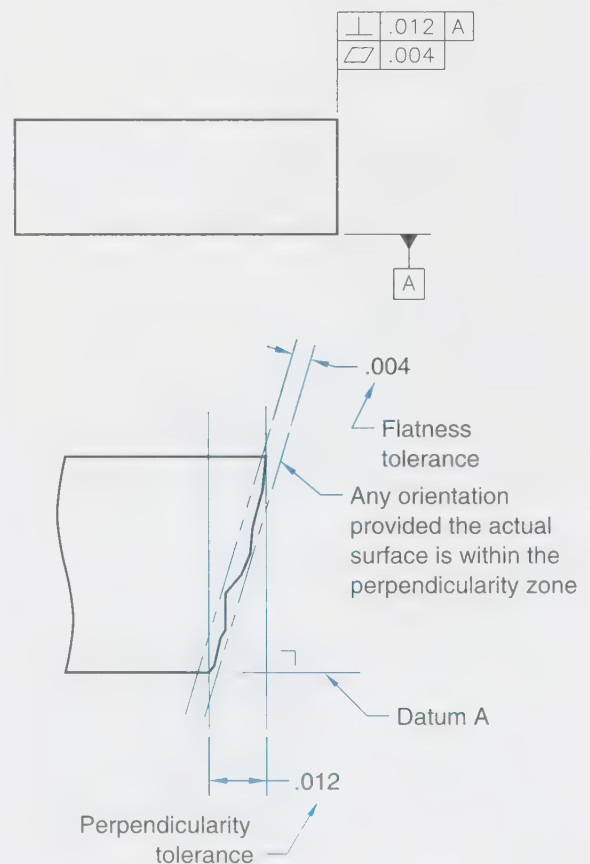
The current standard specifically states that orientation tolerances applied to surfaces also control form. This means that any orientation tolerance on a flat surface also requires the surface flatness to be a value equal to or less than the orientation tolerance. As an example, application of a .006" perpendicularity tolerance also requires flatness within .006". Exception to this requirement is taken when the orientation tolerance is applied to a tangent plane. Any form tolerance applied to a surface must be smaller than any orientation tolerance applied to the same surface.

The standard does not specifically state that orientation tolerances applied to features of size also control form. However, it can be determined that orientation tolerances applied RFS on a feature of size do not control form. When applied RFS, orientation tolerances cannot directly control

form, because form tolerances applied RFS control a different derived feature than is controlled by orientation tolerances. Straightness applied RFS on a feature of size controls the derived median line or derived median plane. Perpendicularity applied RFS on a feature of size controls the axis or center plane (of the unrelated actual mating envelope). Because the two tolerances are controlling two different derived features, it is not possible to show that one must be within the other. So, for features of size, an orientation tolerance applied RFS does not control form. This is explained further in the section titled *Orientation and Axis Straightness*.

Orientation and Flatness on Surfaces

Multiple feature control frames may be applied to an individual feature. See [Figure 7-34](#). This figure shows a perpendicularity tolerance and flatness tolerance applied to the same surface. A perpendicularity tolerance of .012" relative to datum A is specified. In addition, a flatness tolerance of .004" is specified. As stated above, the form tolerance must be smaller than the orientation tolerance when applied to a surface.



Goodheart-Willcox Publisher

Figure 7-34. A form tolerance may be applied to a surface to refine the orientation requirement.

Both feature control frames are placed on one extension line in the given figure. This is not required, but is a common practice when applied in orthographic views. The feature control frames may also be stacked together and connected to the surface using a leader. Application with a leader touching the surface is the method used in solid models and may be used in orthographic views. Stacking of all tolerances for a surface and attachment with a single leader or on a single extension line has the advantage of locating all specifications for a given surface in one place. This helps to ensure that all requirements are easily seen.

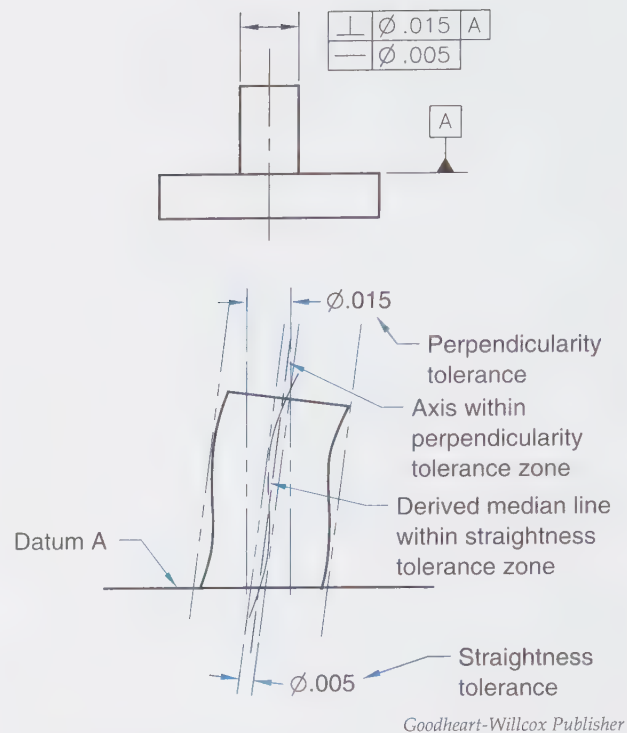
The perpendicularity tolerance in the given figure is shown above the form tolerance. Feature control frames are commonly organized to place the larger tolerance zone first. The subsequent feature control frames indicate tolerances that refine the previous tolerance. This arrangement makes the tolerance requirements easier to read and keeps them in a logical order.

The given figure shows one possible condition for the tolerated surface. There is a .012" wide perpendicularity tolerance zone. The surface may lie anywhere within this zone, provided it is also flat within .004". The figure shows a surface that is flat within .004", and the surface is oriented to extend all the way across the .012" perpendicularity tolerance zone. It is permissible for the tolerance zone boundaries to overlap provided the surface of the part does not violate either zone.

Orientation and Axis Straightness

Figure 7-35 shows a perpendicularity and straightness tolerance applied to a pin. Both tolerances are applied RFS. The perpendicularity tolerance requires the axis of the actual mating envelope lie within a .015" diameter cylinder that is perpendicular to datum A. The straightness tolerance requires the derived median line to be within a .005" diameter cylinder. There is no orientation requirement on the straightness tolerance, other than the requirement that the pin axis meet the .015" perpendicularity tolerance.

It is important to note a small detail in the terminology of the previous paragraph. The orientation tolerance applied RFS is controlling the axis of the actual mating envelope, and the form tolerance applied RFS is controlling the derived median line of the feature. So, they are controlling two different things. If the form variation on the produced part is small relative to the specified tolerance, the difference is relatively insignificant. When form variation on the produced part is larger, the difference becomes more significant.



Goodheart-Willcox Publisher

Figure 7-35. A form tolerance may be applied to a feature of size in combination with an orientation requirement.

Compliance with the orientation tolerance is determined by finding the unrelated actual mating envelope. In this example, the actual mating envelope is a cylinder that best fits to the outside of the tolerated pin. The axis of the actual mating envelope is then checked to ensure that it meets the orientation tolerance requirement.

Compliance with the form tolerance is determined by finding the centers along the length of the pin and determining if they are all within the specified tolerance.

Chapter Summary

- ✓ Orientation tolerances are parallelism, perpendicularity, and angularity.
- ✓ Orientation tolerances may be used to refine the orientation requirements imposed by other tolerances such as position.
- ✓ Surface condition requirements may be refined by showing a form tolerance in addition to the orientation tolerance.
- ✓ A virtual condition exists when an orientation tolerance is applied to a feature of size and an MMC modifier is included in the tolerance specification.
- ✓ Parallelism does not impact size requirements, but it does refine orientation within the size limits.

- ✓ An orientation tolerance applied to a cylindrical feature of size normally requires a diameter symbol.
- ✓ All orientation tolerance specifications must include at least one datum feature reference.
- ✓ Two datum feature references may be used for an orientation tolerance specification.
- ✓ The center plane is controlled when an orientation tolerance is applied to a rectangular feature of size.
- ✓ Orientation tolerances also define allowable surface form variation when applied to a surface.
- ✓ An orientation tolerance may be specified to control the tangent plane.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. Orientation tolerances applied to a surface establish an orientation requirement as well as a ____ tolerance requirement.
 - A. form
 - B. position
 - C. concentricity
 - D. runout
2. A feature control frame must include a minimum of ____ datum feature reference(s) for an orientation tolerance.
 - A. one
 - B. two
 - C. three
 - D. None of the above.
3. An MMB material boundary modifier may be applied to a datum feature reference when _____.
 - A. the primary datum feature is a surface
 - B. the datum feature is a feature of size
 - C. two datum feature references are used
 - D. the tolerance value includes a modifier
4. A parallelism tolerance applied to a flat surface also controls _____.
 - A. size
 - B. position
 - C. flatness
 - D. Both A and C.
5. A tolerance applied to control parallelism of two holes ____ the location tolerance between the holes.
 - A. does affect
 - B. does not affect
 - C. eliminates
 - D. Both A and C.
6. When a perpendicularity tolerance is applied to a surface, the controlled surface must _____.
 - A. lie within the perpendicularity tolerance zone
 - B. lie within any applicable size tolerance
 - C. Neither A nor B.
 - D. Both A and B.
7. A(n) ____ symbol is used to indicate that the orientation tolerance zone is cylindrical.
 - A. MMC
 - B. LMC
 - C. circularity
 - D. diameter
8. An orientation tolerance is applied to a ____ by placing the feature control frame adjacent to the size dimension or on the dimension line.
 - A. surface
 - B. centerline
 - C. feature of size
 - D. None of the above.
9. The virtual condition for a .376" MMC pin with a perpendicularity tolerance of .008" diameter at MMC is _____.
 - A. .008
 - B. .368
 - C. .376
 - D. .384
10. The virtual condition for a .223" MMC hole with a perpendicularity tolerance of .004" diameter at MMC is _____.
 - A. .004
 - B. .219
 - C. .223
 - D. .227
11. An angularity tolerance symbol may be applied on features at _____.
 - A. any angle
 - B. any angle other than 90°
 - C. any angle other than parallel
 - D. acute angles

12. An orientation tolerance on a surface may be specified to control a(n) _____ instead of surface conditions.
 - A. tangent plane
 - B. circumscribing cylinder
 - C. enclosing envelope
 - D. None of the above.
13. A _____ tolerance may be applied to refine (reduce) the allowable surface variation established by an orientation tolerance.
 - A. position
 - B. concentricity
 - C. form
 - D. Both A and C.
24. If a form tolerance is applied to the same surface as an orientation tolerance, the form tolerance must be _____ than the orientation tolerance.
25. Virtual condition is the combined effect of the _____ size and any applicable tolerance.
26. An implied 90° angle is understood to be _____ when a perpendicularity tolerance is applied to perpendicular features.

True/False

14. *True or False?* Size dimensions on a drawing control the tolerance on 90° angles.
15. *True or False?* Two sides of a part must be perfectly flat and parallel when the part is at its least material condition.
16. *True or False?* Perpendicularity applied to a feature of size establishes control of the center plane or axis of the feature.
17. *True or False?* Regardless of the feature type, virtual condition is always larger than the MMC size.
18. *True or False?* Parallelism applied to a flat surface controls flatness to the same value as the parallelism tolerance.
19. *True or False?* Angles measuring 90° must be dimensioned in orthographic views.
20. *True or False?* A perpendicularity tolerance referenced to one datum only controls the orientation of the surface relative to the one datum.
21. *True or False?* Controlling the center plane of a slot with a perpendicularity tolerance does not impose a direct control of the slot side surfaces.
22. *True or False?* An angularity tolerance controls feature orientation and location.

Fill in the Blank

23. Two sides of a part must be perfectly parallel when the feature is at its _____.

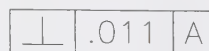
Short Answer

27. Explain why it is sometimes necessary to place an orientation and form tolerance on the same surface.
28. Sketch the symbol for each of the following:
 - A. Angularity
 - B. Parallelism
 - C. Perpendicularity
29. Describe a method for checking parallelism between two flat surfaces.
30. Describe two methods for showing that a perpendicularity tolerance applies to a surface.
31. What is achieved by referencing a secondary datum in an orientation tolerance specification?
32. How is the size of a functional gage calculated for checking perpendicularity of a hole?
33. Define a tangent plane as the term relates to an orientation tolerance.

Application Problems

Some of the following problems require that a sketch be made. All sketches should be neat and accurate. Each problem description requires the addition of some dimensions for completion of the problem. Apply all required dimensions in compliance with dimensioning and tolerancing requirements. Show any required calculations.

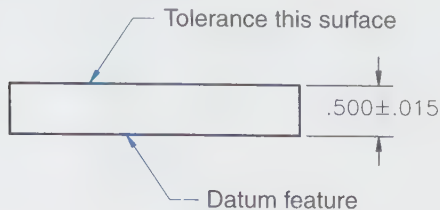
34. Label each segment of the given feature control frame and define the requirements of each segment. Assume the feature control frame is attached to a surface and that the datum feature is a surface.



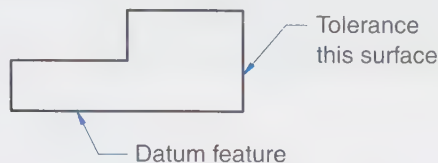
35. Complete a two-line feature control frame that requires parallelism within .015" relative to datum D and a flatness of .008".



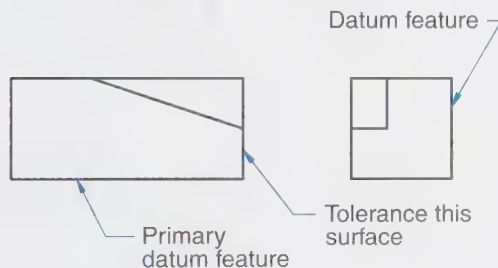
36. Apply a parallelism requirement of .009".



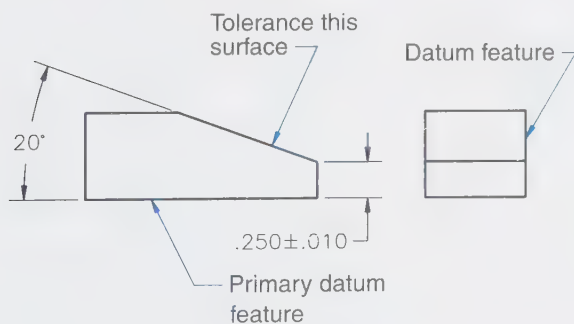
37. Apply a perpendicularity requirement of .007".



38. Apply a perpendicularity requirement of .010" relative to two datums.

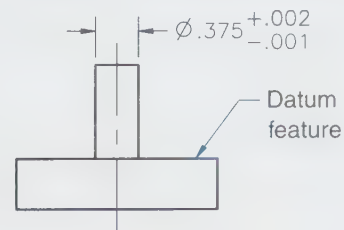


39. Apply an angularity tolerance of .012" relative to two datums.



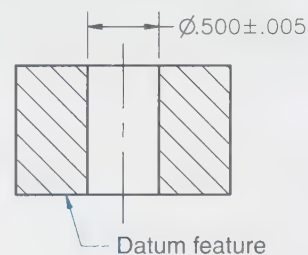
40. Sketch an inspection setup for the part in Problem 39, and describe the requirements for acceptance of the part.

41. Apply a perpendicularity tolerance to control the pin within a .005" cylindrical tolerance zone at MMC relative to the datum surface.



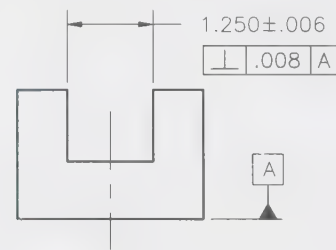
42. Sketch and dimension a theoretical functional gage to check the part in Problem 41. Show your calculations.

43. Apply a perpendicularity tolerance to control the hole within a .007" cylindrical tolerance zone relative to the datum surface.

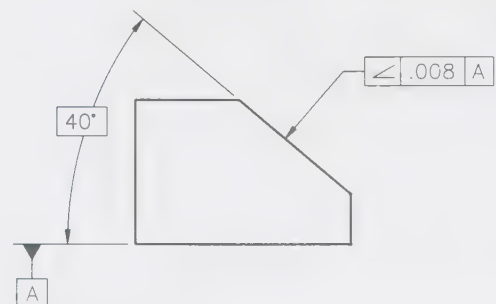


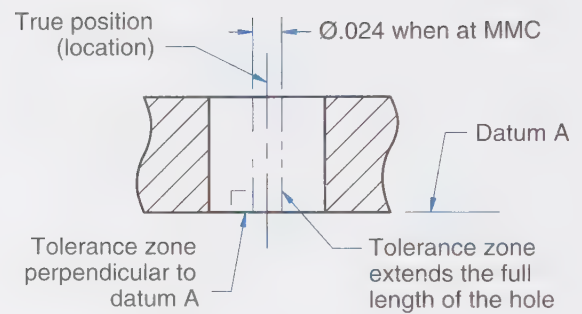
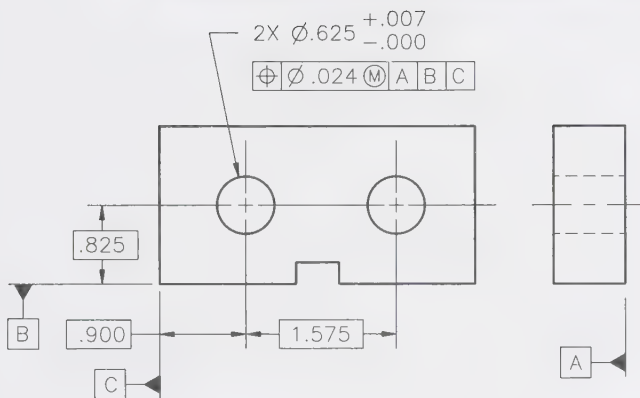
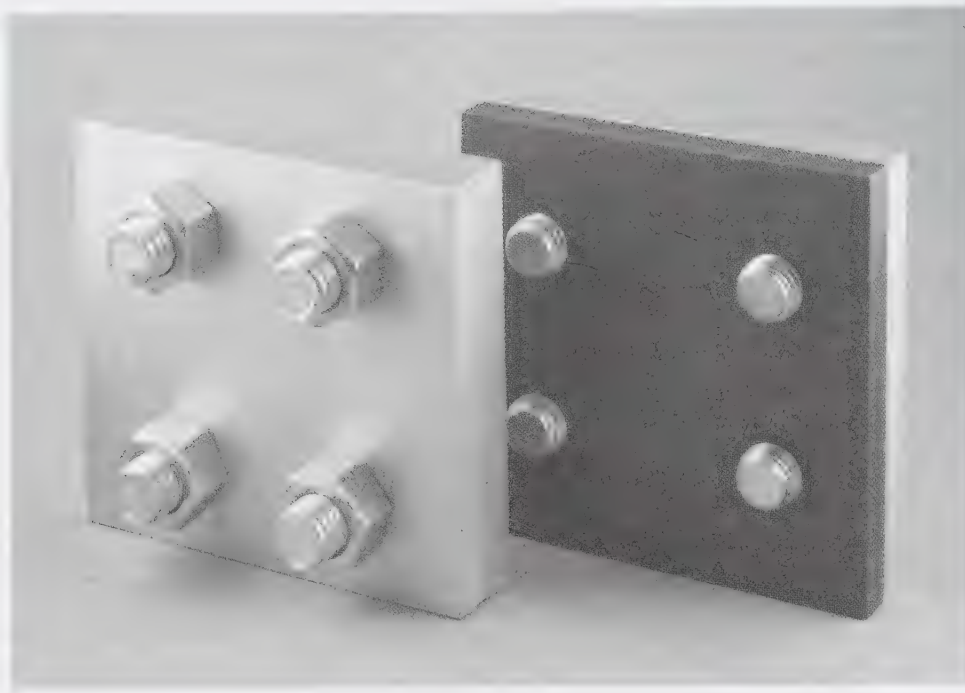
44. Sketch and dimension a theoretical functional gage to check the part in Problem 43. Show your calculations.

45. Sketch one acceptable configuration for a part fabricated to the shown dimensions. Some departure from the nominal dimensions is to be shown.



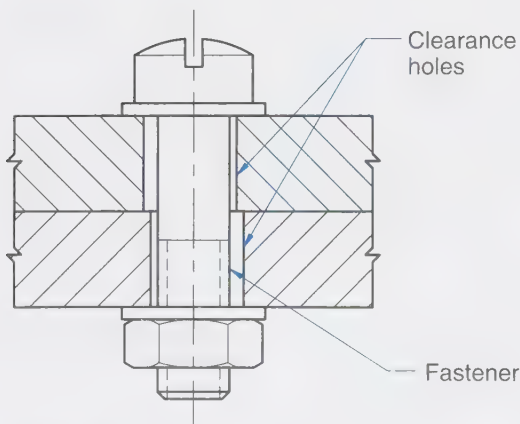
46. Make the necessary changes to apply the shown tolerance to a tangent plane.





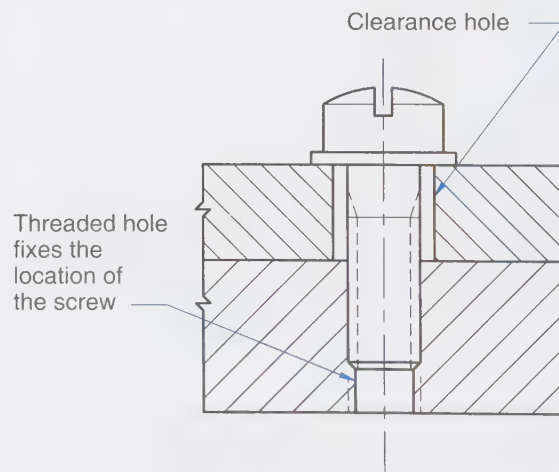
Position tolerance zones

Goodheart-Willcox Publisher



Floating fastener

Goodheart-Willcox Publisher



Fixed fastener

Goodheart-Willcox Publisher

Chapter 8

Position Tolerancing Fundamentals

Objectives

Information in this chapter will enable you to:

- ▼ Complete feature control frames for position tolerances, properly using the diameter symbol, material condition modifiers, and datum references.
- ▼ Sketch the proper location and shape for position tolerance zones.
- ▼ Describe the effect of an MMC, LMC, or RFS modifier on a position tolerance.
- ▼ Provide examples that prove the validity of the MMC concept as it applies to position tolerances.
- ▼ Calculate position tolerances for simple fixed and floating fastener conditions.
- ▼ Calculate the allowable bonus tolerance for a produced part on which a position tolerance is specified at MMC.
- ▼ Use calculation techniques to verify whether produced hole locations meet specified drawing tolerances.
- ▼ Cite advantages of position tolerances when compared to coordinate hole location tolerances.

Technical Terms

additional tolerance
axis method
bidirectional position tolerance
bonus tolerance
boundary method
fixed fastener condition
floating fastener condition
paper gaging
position tolerances
position tolerance zone

projected tolerance zone
surface method
true position

Introduction

Drawings and solid models of production parts include tolerances that define the acceptable amount of variation on dimensions. There must be an allowable variation on all aspects of the part geometry, including the locations for features of size. As an example, the required location for a hole must include a tolerance. One method for defining location tolerances on holes and other features of size is to apply a position tolerance.

Drawings and solid models must define the acceptable amount of variation on sizes and locations because it is not possible to produce perfect parts. This chapter describes how to use position tolerances for specification of allowable location variation.

Throughout this chapter and other chapters of this textbook, allowable feature variation is often explained and illustrated showing the requirement in terms of a feature axis within an allowable tolerance zone. The *axis method* of explaining tolerance requirements may be used for position tolerances applied RFS, MMC, and LMC. An axis of an imperfect feature is defined as being the axis of the unrelated actual mating envelope when the tolerances are applied RFS or MMC. The axis is defined as the axis of the unrelated actual minimum material envelope when the tolerances are applied LMC.

Some explanations show the requirement in terms of an acceptance boundary that the feature surface must not violate. This may be referred to as the *surface method* or *boundary method*. The

surface method is only applicable to tolerances applied with the MMC or LMC modifier.

Provided a feature has no form variations, the axis and surface methods show the same result in terms of measured position variation and part acceptance. When features have form variation, then the two methods may give different results, and the amount of difference is impacted by the amount of form variation in the measured feature. When the results are significantly different, the surface method takes precedence for features toleranced with the MMC or LMC modifier. The surface method is not applicable for features toleranced RFS.

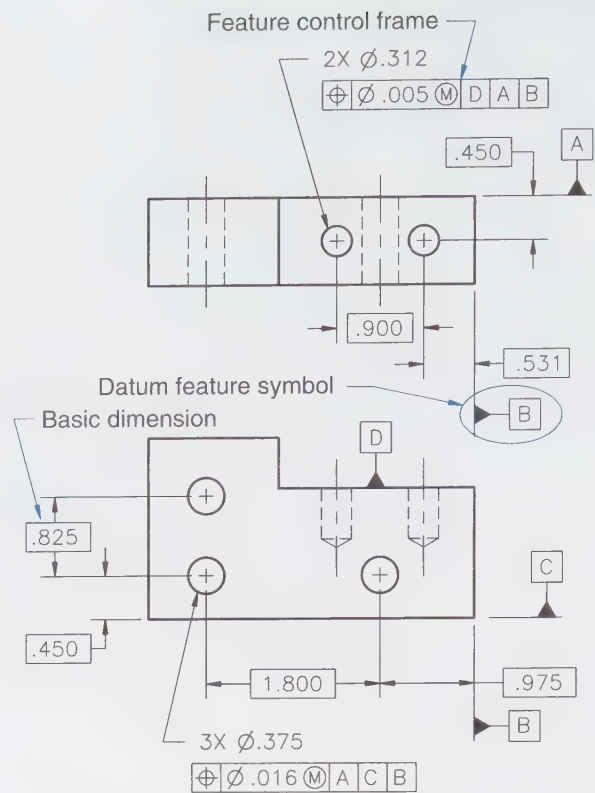
Position

Position tolerances are used to specify the required location accuracy for features of size and bounded features. Features of size typically have a size dimension such as a diameter for a cylinder or a width for a slot. Bounded features are typically enclosed and have a profile tolerance that creates inner and outer boundaries for the feature. Tolerance specification methods make it possible to control the location of as many features as necessary. One feature control frame may be used to show the position tolerance on a single hole, or it may be used to add a tolerance to a pattern of holes.

Any drawing or solid model that includes a position tolerance on holes must include several characteristics. See **Figure 8-1**. Datum features are identified. Basic dimensions are used to define hole locations. Basic dimensions may be shown in a drawing view or they may be defined by the data in a solid model. There is at least one feature control frame that specifies a position tolerance.

Each of these characteristics has a purpose. The datum features serve to locate the datum reference frame from which the hole locations are measured. The basic dimensions define the true positions for the holes. The feature control frame defines the size of the tolerance zone that is located on each of the true positions and references the datum features that are used to create the datum reference frame.

Figure 8-1 shows two feature control frames that define the position tolerances for five holes. The feature control frame in the top view is placed adjacent to a noted size for two holes. Therefore, it gives the position tolerance for both of these holes. The other feature control frame is placed adjacent to the noted size for the three hole pattern. This feature control frame specifies the required position tolerance for each of the three holes.



Goodheart-Willcox Publisher

Figure 8-1. Basic dimensions, datum features, and position tolerance specifications are required to completely define position tolerance requirements.

Both feature control frames in the given figure have a diameter symbol in front of the tolerance value. A diameter symbol in the tolerance specification indicates a round tolerance zone. Omitting the diameter symbol would be incorrect for the given part.

The shape specified for a tolerance zone is determined from the functional requirements for the toleranced feature. Holes are typically given round tolerance zones. Rectangular features are usually given rectangular tolerance zones.

Feature control frames are used to specify position tolerances. The composition of the single-segment (single-line) position tolerance feature control frame defines the tolerance zone shape and size. It also defines the datum reference frame to which the tolerance zone must be located and oriented. Multiple-segment (two-line or three-line) and composite position tolerance feature control frames are more complex. They are explained in the next chapter.

Position tolerances include an implied RFS material condition modifier on the tolerance value if no symbol is shown. When a material condition modifier other than RFS is to apply, either the MMC or LMC modifier must be shown. Material

condition modifiers are appropriate, because position tolerances are only applied to features of size and bounded features. Maximum material condition (MMC) and least material condition (LMC) modifiers permit the tolerance zones to vary according to feature size. This can significantly increase the producibility of the parts.

Feature Control Frame

Information shown in a position tolerance feature control frame must be shown in the correct sequence. See **Figure 8-2**. The tolerance symbol (characteristic) must be shown first. It is followed by the tolerance value, including a diameter symbol if the zone is circular. The tolerance value is followed by a material condition modifier if MMC or LMC is applicable. No modifier is shown if RFS is applicable.

If tolerances are determined using statistical calculations, the statistical symbol (ST) may be inserted in the feature control frame. It is placed after any shown material condition modifier. If the statistical symbol is shown, notations may be added to define any required statistical process control requirements that are applicable to that tolerance.

Datum feature references follow the tolerance value. Implied datum feature references are not permitted on position tolerances. As with all other feature control frames, the datum feature references are read from left to right. The primary datum feature is shown first, followed by the secondary and tertiary datum features.

The requirements regarding datum feature references in positional tolerances have matured during the development of several revisions of the national and international standards. Prior to ANSI Y14.5M-1982, datum feature references were generally shown on position tolerances, although it was not required for all applications. Early dimensioning and tolerancing standards, prior to 1982, permitted implied datums and mixed directly applied tolerances with basic dimensions. If working

with old drawings, these practices may be seen and may also be difficult to interpret.

The utilization of implied datums is no longer permitted because implied datums may cause confusion. Implied datums can result in confusion because a datum reference frame must be assumed. Being required to assume a datum reference frame can introduce inconsistency because two people might assume different datum reference frames. It must be realized that two different datum reference frames will not create the same results when fabricating parts.

The 1982 and 1994 standards did not permit datum feature references to be implied. The ASME Y14.5-2009 standard does not permit implied datums. However, the 2009 standard shows some permissible applications of position tolerances without datum feature references. This does not mean that implied datums are permitted, but they are not needed for certain applications. The applications where datum references are not required are rare exceptions to the general requirement for datum feature references in position tolerances.

Position tolerances within a pattern of features, without datum feature references, is permitted if the pattern of features becomes the primary datum feature to which other features are tolerated. A position tolerance without a datum feature reference is not the same as a position tolerance with implied datum feature references. Implied datum feature references are not permitted.

Position tolerances are only applied to features of size and bounded features. The feature control frame may be connected to the appropriate feature with a leader, it may be placed adjacent to the feature dimension, or it may be placed on the dimension line for the size dimension.

Many position tolerancing applications may be met using the single-segment (single-line) feature control frame already described. More complex applications sometimes require the use of multiple single segments or a composite feature control frame. The format of these feature control frames is shown in **Figure 8-3**. These are explained in the next chapter.

Material Condition Modifier Application

Rule #2 of the current standard requires that all tolerances be assumed to include the RFS material condition modifier. See **Figure 8-4**. Maximum material condition (MMC) and least material condition (LMC) must be shown if they are to apply. Selection of the appropriate modifier depends on the design application.

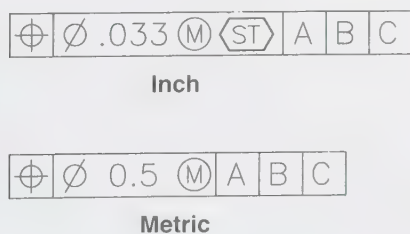


Figure 8-2. A position tolerance feature control frame may be used for inch or metric tolerance values.

\oplus	$\varnothing .024$	(M)	A	B	C
\oplus	$\varnothing .015$	(M)	A		

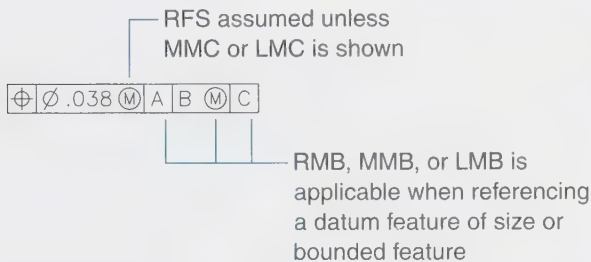
**Two single segments
(Two feature control frames)**

\oplus	$\varnothing .027$	(M)	A	B	C
\oplus	$\varnothing .011$	(M)	A		

**Composite
(One feature control frame)**

Goodheart-Willcox Publisher

Figure 8-3. Single-segment, two-segment, or composite position tolerance feature control frames may be used to achieve the required level of control.



Goodheart-Willcox Publisher

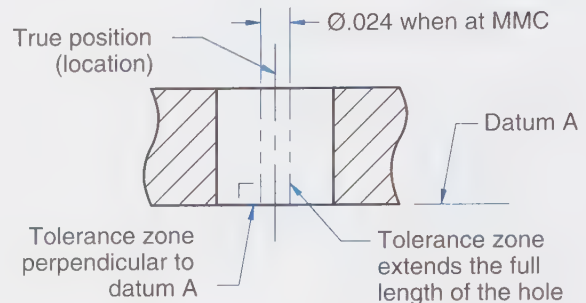
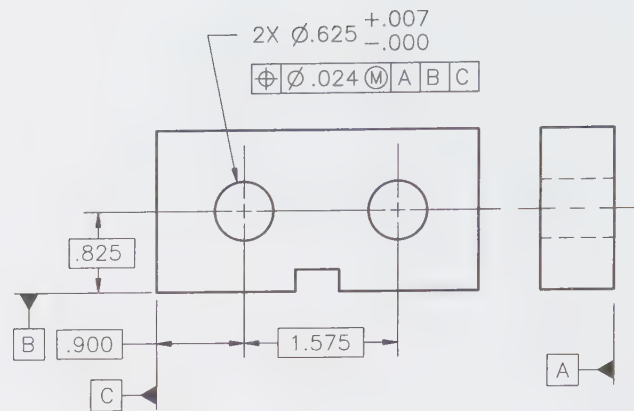
Figure 8-4. Regardless of feature size is assumed on position tolerances. Other applicable modifiers must be shown.

Each modifier has a significantly different effect on the requirements of the specified tolerance. The effect of each modifier is explained in this chapter.

Datum feature references to features of size and any feature that has material boundaries are applicable at a specific material boundary. If no modifier is shown, the datum feature reference is understood to apply regardless of material boundary (RMB). The material condition modifiers used on datum feature references may be different from the material condition modifier used on the tolerance value. As an example, the tolerance value may be applicable at MMC and the datum feature references applicable RMB. Datum feature references to surfaces should not include the MMB or LMB modifier unless the surface has a profile tolerance that establishes material boundaries.

Position Tolerance Zone

A *position tolerance zone* will usually have a defined shape, size, location, and orientation. See [Figure 8-5](#). The shape is generally determined by whether or not a diameter symbol is placed in the feature control frame. The three-dimensional size



Goodheart-Willcox Publisher

Figure 8-5. The position tolerance zone for a hole is cylindrical and extends the full length of the hole unless specified otherwise.

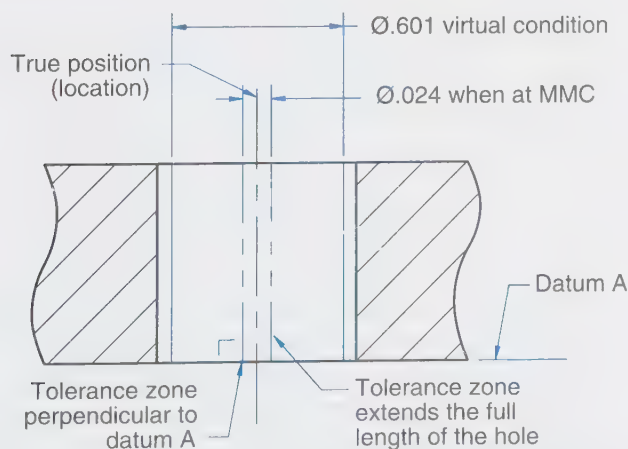
of the tolerance zone is typically defined by the tolerance value shown in the feature control frame and by the length of the hole (or whatever feature of size is being controlled). Basic dimensions and references to datums define the location and orientation requirements for the tolerance zone. In orthographic views, holes are generally defined by drawing convention to be at a basic 90° angle, but orientation (angle) dimensions may be used to specify any required angle. Any angle other than 90° must be dimensioned unless the design is defined by an undimensioned solid model where the digital definition of the geometry establishes feature dimensions.

A typical position tolerance specification and the resulting tolerance zone are shown in [Figure 8-5](#). The shape, size, location, and orientation of the tolerance zone are defined. The tolerance specification has a diameter symbol applied to the .024" tolerance value. This indicates a requirement for the tolerance zone to be a .024" diameter cylinder. The cylindrical zone extends completely through the hole. It terminates flush with the two surfaces. The tolerance zone cylinder is centered on the true position that is defined by the basic dimensions. The tolerance zone on this part must be perpendicular to primary datum A.

The tolerance value includes the MMC modifier, so the tolerance zone diameter must be .024" diameter when the hole is produced at its MMC size. The axis of the hole (meaning the axis of the unrelated actual mating envelope) must be within the tolerance zone.

The preceding paragraph explains the position tolerance requirement in relationship to the axis of the hole. When a position tolerance is applied with the MMC or LMC modifier, the tolerance requirement may also be explained in relationship to the surface of the hole. See [Figure 8-6](#). The hole from [Figure 8-5](#) is shown at a larger scale, and a boundary known as the virtual condition (Vc) is shown. The surface of a manufactured hole must not violate the virtual condition that is based on the hole MMC and the position tolerance. The virtual condition for a hole is the MMC size minus the position tolerance. In the given example, the virtual condition is .601" diameter.

When the MMC or LMC modifier is applied to the tolerance, the axis and surface methods may give different results when measuring holes that have form variation (imperfect cylinders). When there is a difference, the surface method takes precedence. Even though the surface method takes precedence, the axis method is generally used throughout industry for explaining position tolerances. The axis method is also most often used for measuring feature locations because of the ease of performing calculations. Provided the form of the measured features is relatively accurate, the differences between the axis and surface methods are generally insignificant and the axis method is adequate. The surface method cannot be used when the tolerance is applied RFS and the axis method must be used.



Goodheart-Willcox Publisher

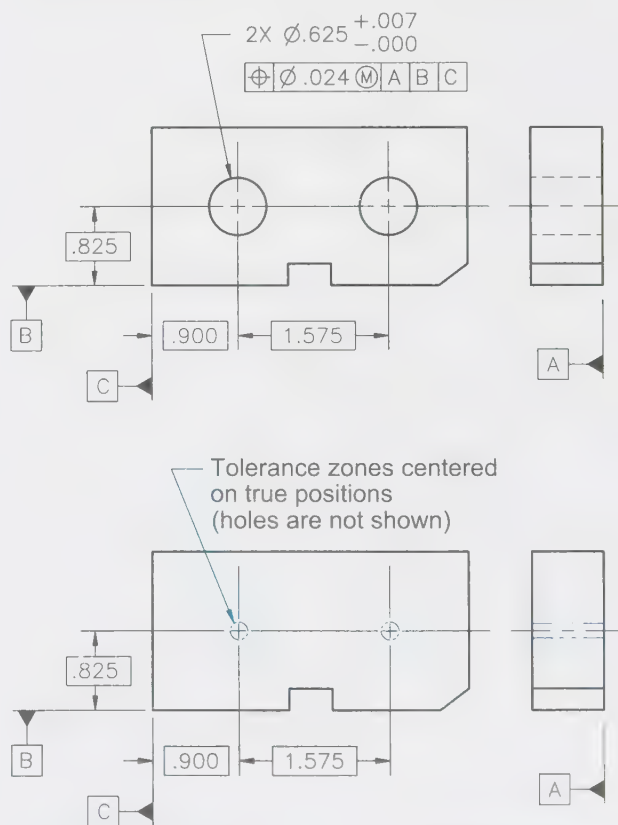
Figure 8-6. The axis of a hole is at the center of the unrelated actual mating envelope.

Basic Dimensions and True Positions

Basic dimensions must be used to define the locations of features to which position tolerances are applied. See [Figure 8-7](#). The given figure shows two holes located by three basic dimensions. Basic dimensions are theoretically perfect and have no tolerance on them.

The basic dimensions define the true positions for the two holes. **True position** is a theoretically exact location and orientation defined by basic dimensions. The true position is exact and is related to the referenced datums, but a true position does not indicate the hole is required to be perfectly located. There must always be an allowable amount of position tolerance. The true position is where it is desired for the hole to be and the specified position tolerance zone is centered on the true position. The axis of the hole may be located anywhere within the tolerance zone that is centered on the true position.

[Figure 8-7](#) shows two horizontal dimensions chained together to locate the true positions of the two holes. There would be no difference in the true positions if both holes were dimensioned with baseline dimensions extending from the datum feature. There would not be a difference because



Goodheart-Willcox Publisher

Figure 8-7. Tolerance zones for holes are centered on the theoretically true positions for the holes.

there is no tolerance accumulation on basic dimensions. There is no tolerance applied to the basic dimension itself; the position tolerance is applied to the feature.

Relationship to the Datum Reference Frame

Position tolerances must include datum feature references except when the toleranced features act as the primary datum from which other features are located. This exception is explained in more detail in later sections of this chapter. The datum feature references establish the datum reference frame from which measurements must be made. See [Figure 8-8](#). A datum reference frame provides a known origin and orientation for measurements. Review Chapter 6 for additional information about datum reference frames.

Dimensions are applied with consideration given to referenced datums. The given part shows a position tolerance in which datum features A, B, and C are referenced. The hole pattern is dimensioned to locate it relative to these datums. In a solid model where no dimensioned views are created, the hole locations relative to the datum reference frame are established by the digital data in the model.

All location measurements for the two holes are made relative to the datum reference frame,

and not relative to the part surfaces. There is a difference. The datum reference frame is a perfect coordinate system. The part surfaces have variations. Depending on the magnitude of the part variations, measurements from the datum reference frame could be significantly different from measurements from the part surfaces.

The exaggerated variations on the shown part indicate the importance of the datum reference frame. When working relative to the datum reference frame, all measurements are in a single coordinate system. Being able to remain in one coordinate system is unlikely when working off the imperfect feature surfaces. Because of angular variation between datum features B and C, measurements made off the two surfaces may not be perpendicular to one another. If the measurements are not perpendicular, then an accurate check of the dimensions cannot be made.

Effect of Material Condition Modifiers

The three material condition modifiers have distinctly different effects when applied to a position tolerance. One of the modifiers is optimum for any given design application. Selection of the correct modifier is relatively easy when the modifier effects are clearly understood.

Regardless of Feature Size

No regardless of feature size (RFS) modifier is needed because RFS is implied. See [Figure 8-9](#). The assumed RFS modifier indicates that the specified tolerance value is to remain constant, regardless of the produced size of the toleranced part feature.

A hole specification and position tolerance are shown in the given figure. Limits of size for the hole are .500" and .510" diameter. The position tolerance is specified as .025" diameter RFS.

Any hole produced within the limits of size is permitted a position tolerance of .025" diameter. The size of the hole has no effect on the allowable position tolerance because the tolerance is applicable RFS.

Two examples of produced holes are shown. One hole is produced at .500" diameter, the maximum material condition for the hole. The position tolerance for this hole is .025" diameter. The other hole is produced at .510" diameter, the least material condition for the hole. The position tolerance for this hole is also .025" diameter.

RFS is typically used for applications where holes have zero clearance or press fits. The allowable

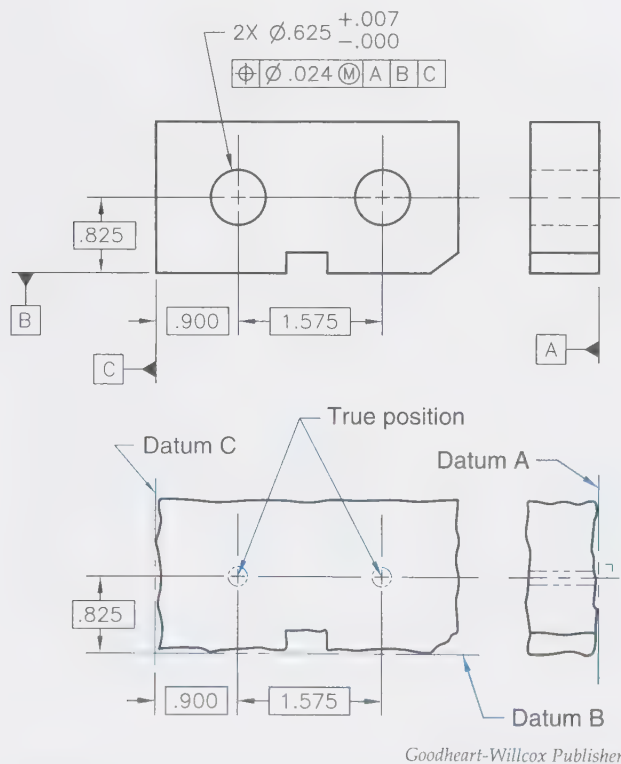
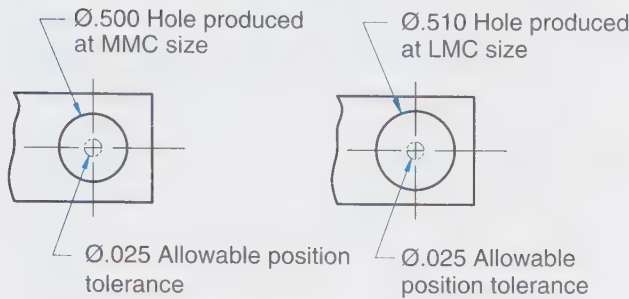


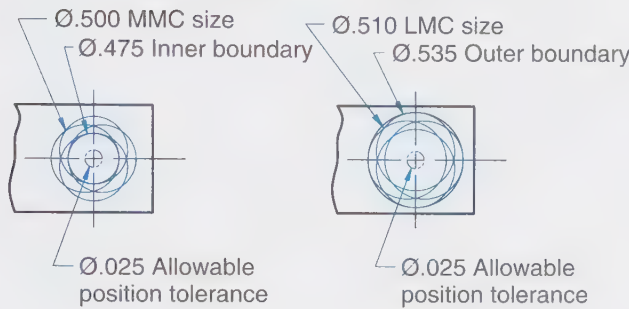
Figure 8-8. Datum feature references in a position tolerance specification require that measurements be made relative to a datum reference frame.



Hole specification and position tolerance



Produced hole sizes and allowable position tolerances



Inner and outer boundaries

Goodheart-Willcox Publisher

Figure 8-9. The position tolerance zone is not affected by the produced feature size when the RFS modifier is applicable to the tolerance specification.

diameter variation on a press fit hole does not provide any additional position tolerance. Therefore, RFS is required. RFS may also be used for clearance holes that locate other parts.

Two boundaries are created by a position tolerance that is applicable RFS on a hole. There is an inner boundary that is equal to the MMC hole size minus the position tolerance. An outer boundary is created by the LMC hole size plus the position tolerance.

For the inner boundary, the given figure shows four circles at the MMC size and at four different locations at the allowable limits of the position tolerance relative to the same true position. An inscribing circle is drawn representing the inner boundary. Any hole produced within the size and position tolerance will not violate this boundary.

For the outer boundary, the given figure shows four circles at the LMC size and at four different locations at the allowable limits of the position tolerance relative to the true position. A circumscribing

circle is drawn representing the outer boundary. Any hole produced within the size and position tolerance will not violate this boundary.

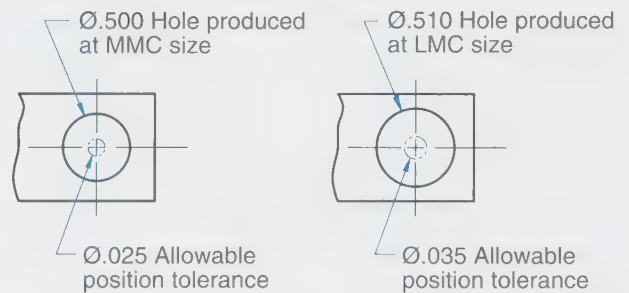
The inner and outer boundaries for position tolerances applied RFS are not acceptance criteria for produced holes. The boundaries are important for design considerations where edge distances, potential effect on assembly variation, or other variation effects need to be analyzed.

Maximum Material Condition

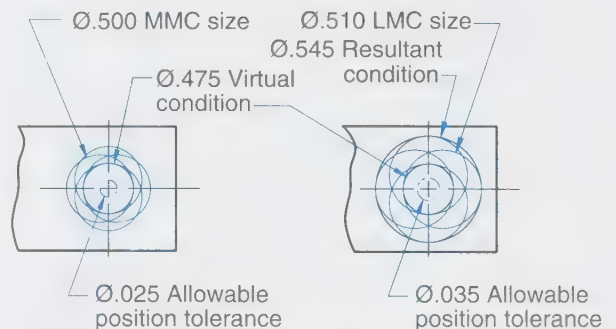
The maximum material condition (MMC) modifier is a letter "M" inside a circle and is shown following the tolerance value. See [Figure 8-10](#). The shown specification is for a hole. The MMC modifier indicates that the specified tolerance value applies only when the hole is actually produced



Hole specification and position tolerance



Produced hole sizes and allowable position tolerances



Virtual and resultant condition boundaries

Goodheart-Willcox Publisher

Figure 8-10. The position tolerance zone increases in size when the MMC modifier is applied to the tolerance specification and the produced feature size departs from MMC.

at MMC. If the hole is produced at any allowable size other than MMC, the position tolerance value increases an amount equal to the size departure from MMC. When using the axis method of verification, the allowable increase in positional tolerance is based on the size of the unrelated actual mating envelope of the hole instead of the size of the hole. Because the following examples have no form variation, the size of the hole and the size of the unrelated actual mating envelope are the same. To simplify the explanations, the size of the unrelated actual mating envelope and the produced feature size are assumed to be equal.

A hole specification and position tolerance are shown in the given figure. Limits of size for the hole are .500" and .510" diameter. The position tolerance is specified as .025" diameter at MMC.

Any hole produced at the exact MMC size of .500" diameter is permitted a position tolerance of .025" diameter. Any hole produced at a larger diameter is permitted a larger amount of position tolerance. For every .001" diameter increase in hole size, there is an allowable .001" increase in the diameter of the position tolerance zone.

Two examples of produced holes are shown. One hole is produced at .500" diameter, the maximum material condition for the hole. The position tolerance for this hole is .025" diameter. The other hole is produced at .510" diameter, the least material condition for the hole. The position tolerance for this hole is increased to .035" diameter.

The MMC modifier does not permit violation of the limits of size. The example hole must not be produced at a size smaller than .500" or greater than .510".

The MMC modifier is typically used for clearance hole applications. An example is a clearance hole through which a bolt is passed. It makes sense to allow the position tolerance to increase as hole size increases on such applications. If a clearance hole is made larger, it may be off true position by a greater amount and still let the bolt pass through. It is logical that the larger a clearance hole is, the more position variation that can be allowed and the fastener still pass through the hole.

Two boundaries are created by a position tolerance that is applicable at MMC on a hole. There is a *virtual condition* (an inner boundary for a hole) that is equal to the MMC hole size minus the specified position tolerance. A *resultant condition* (an outer boundary for a hole) is created by the LMC hole size plus the allowable position tolerance.

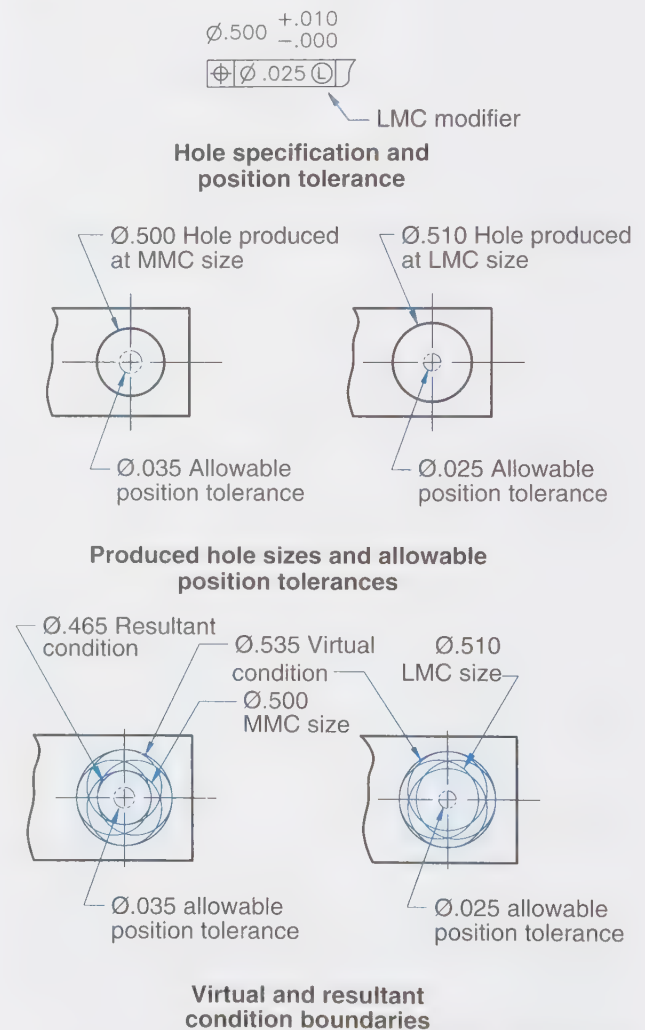
The given figure shows four circles at the MMC size and at four different allowable locations on the same true position. An inscribing circle is

drawn representing the virtual condition. Any hole produced within the allowable size and position tolerance will not violate the virtual condition.

The given figure shows four circles at the LMC size and at four different locations at the allowable limits of the position tolerance relative to the true position. It can be seen that this condition does not violate the virtual condition. Additionally, a circumscribing circle is drawn representing an outer boundary known as the resultant condition. Any hole produced within the allowable size and position tolerance will not violate the virtual condition or the resultant condition.

Least Material Condition

The least material condition (LMC) modifier is a letter "L" inside a circle. See **Figure 8-11**. The shown specification is for a hole. The LMC modifier



Goodheart-Willcox Publisher

Figure 8-11. The position tolerance zone increases in size when the LMC modifier is applied to the tolerance specification and the produced feature size departs from MMC.

indicates that the specified tolerance value applies only when the controlled feature is actually produced at the LMC size. If the controlled feature is produced at any allowable size other than LMC, the position tolerance value increases an amount equal to the size departure from LMC. This is similar to the MMC concept, except it uses the LMC size as the size to which the original tolerance value is applied. When using the axis method of verification, the allowable increase in positional tolerance is based on the size of the unrelated actual minimum material envelope of the hole instead of the size of the hole. For a hole, the unrelated actual minimum material envelope is the smallest circumscribing cylinder that is in the material. Because the form of each hole shown in the following examples has no variation, the size of the hole and the size of the actual minimum material envelope are the same.

A hole specification and position tolerance are shown in the given figure. Limits of size for the hole are .500" and .510" diameter. The position tolerance is specified as .025" diameter LMC.

Any hole produced at the exact LMC size of .510" diameter is permitted a position tolerance of .025" diameter. Any hole produced at a smaller diameter is permitted a larger amount of position tolerance. For every .001" diameter decrease in hole size, there is an allowable .001" increase in the diameter of the position tolerance zone.

Two examples of produced holes are shown. One hole is produced at .510" diameter, the least material condition for the hole. The position tolerance for this hole is .025" diameter. The other hole is produced at .500" diameter, the maximum material condition for the hole. The position tolerance for this hole is increased to .035" diameter.

The LMC modifier is typically used for applications where edge distance or remaining material is of concern. Examples are a clearance hole centered in a boss or a structural hole near the edge of a plate. Another example is a hole in a casting where the position of the hole must be adequately controlled to ensure subsequent machining operations will have material for removal.

Two boundaries are created by a position tolerance that is applicable at LMC on a hole. There is *virtual condition* (outer boundary for a hole) that is equal to the LMC hole size plus the position tolerance. The virtual condition for LMC is actually a boundary inside the material, and this boundary must not be violated by any segment of the hole. A *resultant condition* (an inner boundary for a hole) is also created and its size is calculated using the MMC hole size minus the allowable position tolerance. The

resultant condition of a hole (for LMC applications) is the smallest boundary that must not have any material inside it.

The given figure shows four circles at the LMC size and at four different allowable locations on the same true position. A circumscribing circle is drawn representing the virtual condition. Any hole produced within the size and position tolerance will not violate the virtual condition.

The given figure shows four circles at the MMC size and at four different locations at the allowable limits of the position tolerance relative to the true position. An inscribing circle is drawn representing the resultant condition. Any hole produced within the allowable size and position tolerance will not violate this boundary.

Proof of the MMC Concept

The MMC modifier should be used in a majority of clearance hole applications where the main concern is free assembly of the parts. Application of MMC permits greater freedom in how the part is produced. However, if location accuracy for mating parts is a concern, the effects of larger holes and larger position tolerances may make the application of MMC undesirable. Where mating part location is important, applying tolerances RFS may be appropriate.

Application of an MMC modifier on a position tolerance permits an increase in location tolerance as the hole size is increased. This provides additional manufacturing flexibility during the production of the part.

When the MMC modifier is on the tolerance, the machinist or fabrication planner may determine how to best utilize the total permitted tolerance. It may be determined that it is best to drill a hole near its least material condition. This allows a maximum amount of position tolerance. In some situations, it may be determined that it is best to work near the MMC size, and try to achieve the specified position tolerance. Working near the MMC limit permits more latitude for rework of the part if an error is made. For some very expensive parts, allowing some remaining material for rework can be important to avoid scrapping a part because of small discrepancies.

Understanding how feature size affects position tolerance requirements supports making decisions regarding the utilization of the MMC modifier on position tolerances. The following explanation shows how feature size can affect the allowable position tolerance.

Figure 8-12 shows two simple plates. One has two pins, and the other has two clearance holes. It

is not necessary to know the dimensional information for the two parts. The geometry and relative sizes can be used to see how size affects location requirements.

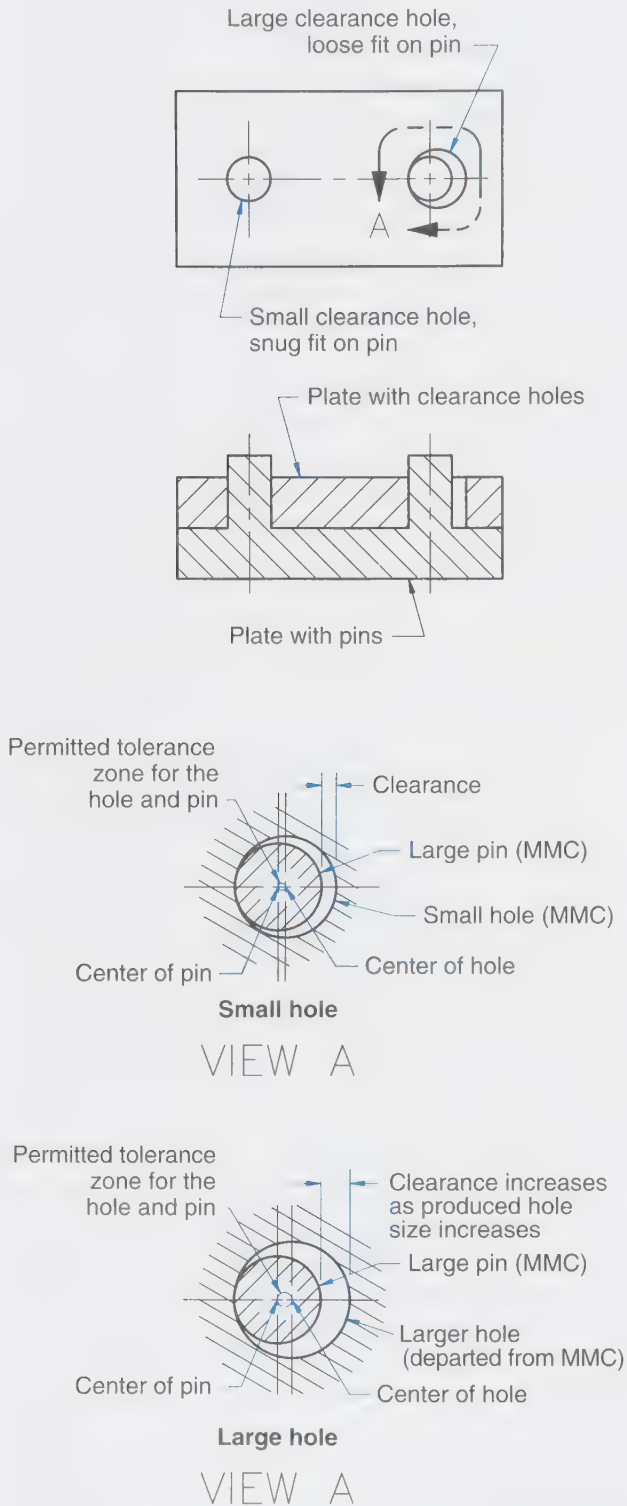


Figure 8-12. A hole with a large amount of clearance can tolerate a larger location variation from true position than can a close-fitting hole.

The only functional requirement known for the given parts is that the plate with clearance holes be able to fit over the two pins. To keep the explanation simple, it is assumed that the plate with the pins is produced as a perfect part. The effects of varying the clearance holes in the mating plate are shown in the given figure.

One of the clearance holes has a snug fit on its pin (there is no visible clearance in the figure). The second clearance hole is larger, and has some clearance between it and the pin. There are two detail views of the second hole.

The first detail view shows an extreme possible position for the clearance hole when the hole is at its smallest limit of size (the MMC size). Only a small amount of clearance exists between the hole and the pin and therefore only a small amount of position tolerance is possible. The acceptable location tolerance for the hole is directly related to the amount of clearance between the pin and the hole.

The second detail view shows an extreme possible position for the clearance hole when the hole is at the maximum limit of size (departed from MMC). The amount of clearance between the hole and the pin has increased because of the increase in hole size. The increased clearance permits a greater amount of location variation.

Increased hole size in the above explanation resulted in an increase in the amount of permitted location tolerance. The characteristics described show that departure from MMC does permit increased location tolerance when clearance fits are used in an assembly. This provides proof that the MMC concept is valid.

Application of an MMC modifier on a tolerance value has a well-defined impact. *When the MMC modifier is applied to a tolerance value, the permitted position tolerance increases by exactly the same amount as the departure of the unrelated actual mating envelope size from MMC.*

Tolerance Calculation

Tolerance values should always be calculated. Calculation is the only means of making sure the tolerances will always result in produced parts that will assemble properly. There are multiple approaches to calculating tolerances. The limit stack method (worst case calculation) ensures that all parts will assemble even if all parts are made at the very worst limit of the tolerance values. Another method is to find the square root of the sum of the squares for all tolerance values. This method assumes that variation is never at the worst case on all parts. Other statistical methods may be used

including computer simulation of fabrication variation combined with assembly sequences and locating constraints. The scope of the explanations in this text primarily addresses the limit stack approach.

Two simple formulas are all that must be remembered to complete the calculations for many common tolerancing applications where a limit stack analysis is appropriate. These two formulas and their proper use are defined in the following paragraphs. These formulas are used to calculate tolerances for the assembly of parts in either of two fastener conditions. These conditions are the floating fastener condition and the fixed fastener condition.

Floating Fastener Condition

A *floating fastener condition* exists when a fastener passes through multiple holes and none of those holes fix the location of the fastener. A floating fastener condition exists when all the holes have a diameter larger than the fastener. See **Figure 8-13**. Two plates are shown; each has a clearance hole that is larger than the bolt that passes through them. Because the holes are larger than the bolt, the bolt is free to float within them.

When designing parts that assemble in a floating fastener condition, it is common to use the same size clearance hole in each of the parts. When using the same size holes, the following simple formula may be used to calculate a position tolerance:

$$T = H - F$$

T = Tolerance (applied to each hole)

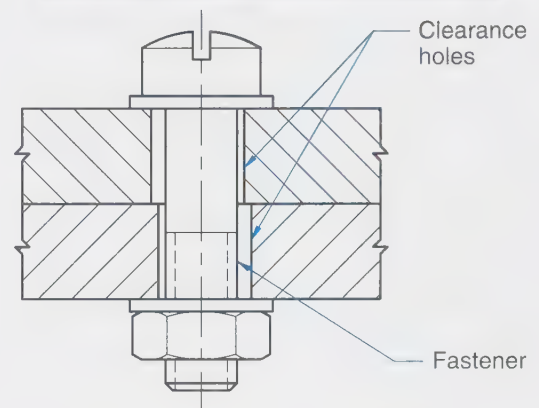
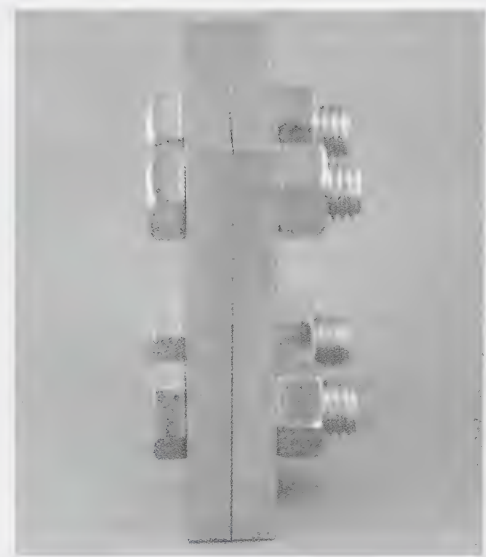
H = Hole (MMC size)

F = Fastener (MMC size)

Care must be taken to only use this formula for the holes in a floating fastener application and not for holes in a fixed fastener application. The calculated tolerance value (T) is the maximum position tolerance that may be applied to the holes on each of the parts.

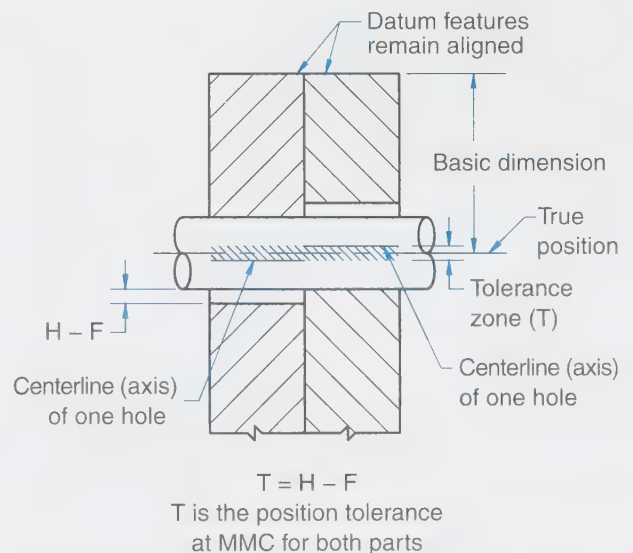
The hole and fastener sizes used in the calculation must be the MMC sizes of each feature, because it is the MMC sizes that result in the worst condition fit (the least clearance) between the two features. Because the position tolerance value is determined using the MMC sizes for the parts having a clearance fit, the tolerance may be specified with the MMC modifier if the relative positions of the mating parts are not a primary concern.

Figure 8-14 shows information that validates the floating fastener formula. The figure shows two plates. The edges of the plates are aligned (assuming they are the referenced datum features) and held in a fixed location relative to one another.



Goodheart-Willcox Publisher

Figure 8-13. A floating fastener condition exists when a bolt or shaft passes through two clearance holes.



Goodheart-Willcox Publisher

Figure 8-14. The floating fastener formula calculates the allowable position tolerance based on the clearance between the hole and fastener.

A hole is shown in each of the plates. The two holes are equal in size. A shaft is passed through the two holes. The two holes and the shaft are all assumed to be at MMC.

The shaft is shown centered on the true positions for the holes. The holes are offset to the maximum extent possible without moving or interfering with the shaft. Offsetting the holes in opposite directions locates the centerlines (axes) of the holes on opposite sides of the true position axis. The distance between the two offset centerlines is equal to T , which is equal to $H - F$.

The condition shown is the worst case. Any other position of either hole would be better than the condition shown. Because the tolerance $T = H - F$ works for the worst case condition, it will work for all others.

Example Calculation Problem:

Required: Calculate the position tolerance for a floating fastener application.

Given information:

Specified hole diameter: $.328" \pm .004"$

Fastener diameter: $.312"$ MMC

Solution:

$$T = H - F$$

$$T = .324" - .312"$$

$$T = .012"$$

The solution of a floating fastener problem requires that at least two parameters be known. If there is only one known parameter, such as the size of the fastener, then either a tolerance or a hole size must be selected. Usually, a standard hole size may be selected that results in a position tolerance that is large enough to be practical for fabrication.

The floating fastener formula can easily be used to solve for a hole size when the fastener and a preselected tolerance are known. Simple mathematics is used to solve for the hole size.

$$T = H - F$$

$$T + F = H - F + F$$

$$T + F = H$$

$$H = T + F$$

Fixed Fastener Condition

A *fixed fastener condition* exists when a fastener passes through multiple holes and one of those holes fixes the location of the fastener. The fastener location can be fixed by threads, a press fit, a taper, or any other feature that prevents movement of the fastener.

Only one part is permitted to fix the fastener location. The other parts include clearance holes. See **Figure 8-15**. Two plates are shown. One has a threaded hole. The other has a clearance hole. The fastener has a location that is fixed by the threaded hole. If the threaded hole is produced somewhere off true position, then the fastener will be pulled off true position with the hole.

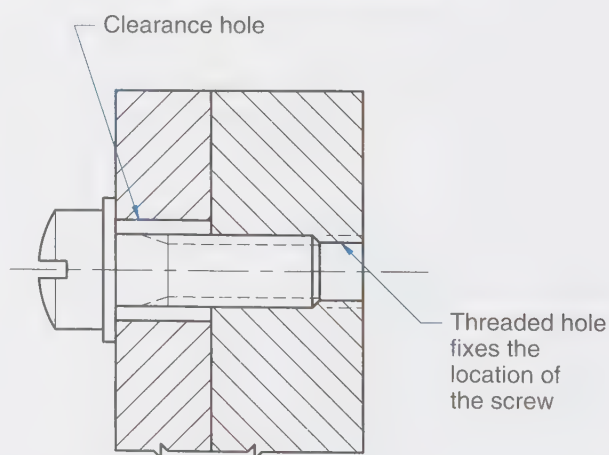
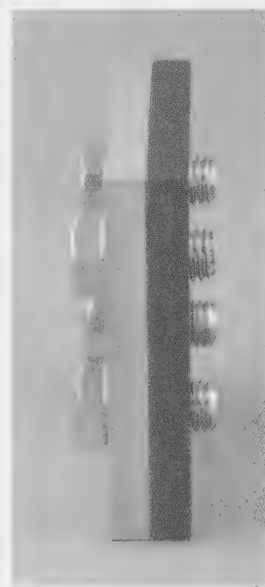
When designing parts that assemble in a fixed fastener condition, the following simple formula may be used to calculate a position tolerance:

$$T = (H - F)/2$$

$$T = \text{Tolerance (applied to each hole)}$$

$$H = \text{Hole (MMC size)}$$

$$F = \text{Fastener (MMC size)}$$



Goodheart-Willcox Publisher

Figure 8-15. A fixed fastener condition exists whenever the location of the fastener is determined by one of the parts.

Note

This formula assumes the tolerance on the fixed fastener feature will be applied using a projected tolerance zone. (Further explanation is given later in this chapter.)

Care must be taken to use this formula for the holes in a fixed fastener application. The calculated tolerance value is the number that is applied to the holes on each of the parts. This formula and the resulting values should only be used if the tolerance applied to the fixed fastener feature is specified as a projected tolerance zone or the clearance hole goes through a thin part.

Figure 8-16 shows information that validates the fixed fastener formula. The figure is simplified for illustration purposes and shows a tolerance zone within the threaded hole. In practice, the tolerance zone for the threaded hole should be specified to project into the clearance hole. Projected tolerance zones are explained later in this chapter.

The figure shows two parts. The edges of the two parts (noted as datum features in the figure) are aligned and held in a fixed location relative to one another. A threaded hole is shown in one part; a clearance hole is shown in the other. A fastener is passed through the clearance hole and threaded into the second part. The fastener and the threaded hole have the same centerline (axis) because screw

threads are self-centering. The clearance hole and the fastener are shown to be at their MMC size.

Neither the fastener nor the clearance hole are shown centered on the true positions for the holes. The threaded hole and the clearance hole are offset in opposite directions. Because moving the threaded hole to one side also moved the fastener, the clearance hole could not be moved very far before one side of the clearance hole made contact with the fastener. In effect, the permitted movement of both the threaded and clearance hole depends on the clearance created by the clearance hole diameter.

The diametral difference between the clearance hole and the fastener is the total amount of position tolerance that may be split between the threaded hole and the clearance hole. When evenly divided, the two features may move equal distances on opposite sides of the true position centerline, just as the figure shows.

Example Calculation Problem:

Required: Calculate the position tolerance for a fixed fastener application.

Given information:

Specified hole diameter: $.296" \pm .003"$

Fastener diameter: $.250"$ MMC

Solution:

$$T = (H - F)/2$$

$$T = (.293" - .250")/2$$

$$T = .043"/2$$

$$T = .0215"$$

Just as for a floating fastener condition, the solution of a fixed fastener problem requires that at least two parameters be known. The fixed fastener formula may easily be used to solve for a hole size when the fastener and a preselected tolerance are known. Simple mathematics is used to solve for the hole size.

$$T = (H - F)/2$$

$$2T = H - F$$

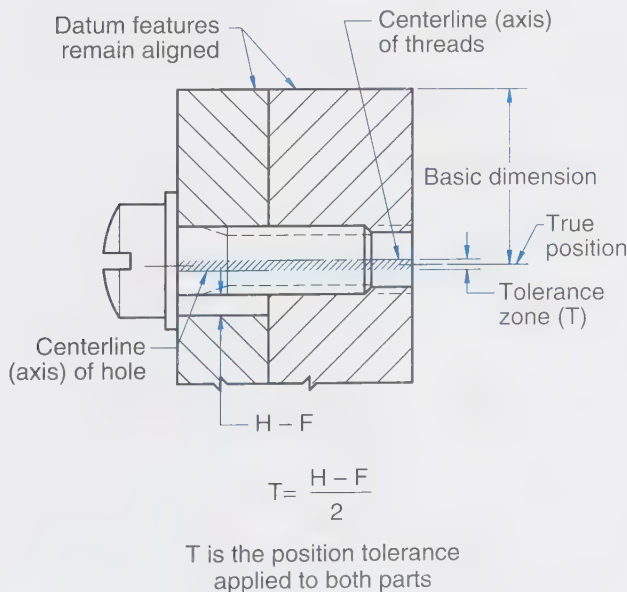
$$2T + F = H - F + F$$

$$H = 2T + F$$

Bonus Tolerances (Additional Tolerance)

Bonus tolerance is not a term found in the ASME Y14.5-2009 standard, but it is a term commonly used in industry. Bonus tolerance is called **additional tolerance** in the standard.

The previously explained formulas, when used for calculating position tolerances, result in a numerical value that is based on a fastener passing through the holes. Position tolerances may also be



Goodheart-Willcox Publisher

Figure 8-16. With the shown fixed fastener formula, the difference in diameters between the clearance hole and the fastener is divided evenly between the position tolerances for the two parts.

calculated for the purpose of protecting the edge distance or spacing between holes. The calculated value is typically the amount of tolerance shown in the feature control frame.

Depending on the design function, the tolerance will be specified as applicable RFS, at MMC, or at LMC. When applicable RFS, no material condition modifier is shown. When the tolerance is applicable at MMC or LMC, the feature control frame will include the MMC or LMC material condition modifier as applicable.

When the MMC modifier is shown, the tolerance value in the feature control frame only applies when the controlled feature is produced at the MMC size. At any size other than MMC, the allowable tolerance zone is increased. The allowable increase in the tolerance is directly related to the departure of the controlled feature from its MMC size. The allowable increase in the tolerance is known as the *bonus tolerance* or *additional tolerance*.

The allowable position tolerance for a produced feature is equal to the specified position tolerance plus the applicable bonus tolerance. When using the surface (boundary) method, the amount of bonus tolerance depends on the actual produced size of the feature. When using the axis method, the size of the actual mating envelope is used to calculate bonus tolerance. The size of the actual mating envelope for a hole is the size of an inscribing cylinder. Therefore, the size of the actual mating envelope is affected by feature size and form variations. Because the form of each feature shown in the following illustrations is drawn without variation, the size of the hole and the size of the actual mating envelope are the same. It is possible to calculate the allowable bonus tolerance for all of the possible produced sizes, but the exact allowable bonus for a particular feature is not known until the feature is produced.

Figure 8-17 shows the effect of the MMC modifier on the position tolerance for a hole. Limits of size for the hole are .500" and .506" diameter. The specified position tolerance of .035" diameter applies when the hole is at its MMC size of .500" diameter. The chart shows that for every .001" increase in the hole diameter, there is a .001" increase in the allowable position tolerance.

The allowable position tolerance can easily be calculated. The allowable tolerance is equal to the sum of the specified tolerance and the bonus tolerance. The bonus tolerance is determined by finding the difference between the produced hole size and the MMC hole size.

The figure shows how the allowable tolerance is calculated for a hole produced at a .503" diameter.

The difference between the produced size (.503") and the specified MMC size (.500") is found to determine the bonus tolerance (.003"). The bonus tolerance (.003") and the specified position tolerance (.035") are added together to determine the allowable position tolerance (.038"). Another way to calculate the allowable position tolerance for a hole is to find the difference between the produced hole size and the virtual condition size. See **Figure 8-17**.

Position tolerances applied to external features are affected by modifiers in the same manner as the tolerances on internal features. See **Figure 8-18**. The pin in the given figure has limits of size from .375" to .379" diameter. A position tolerance of .028" diameter at MMC is applied to the pin.

When the pin is produced at the MMC size of .379" diameter, the allowable position tolerance is

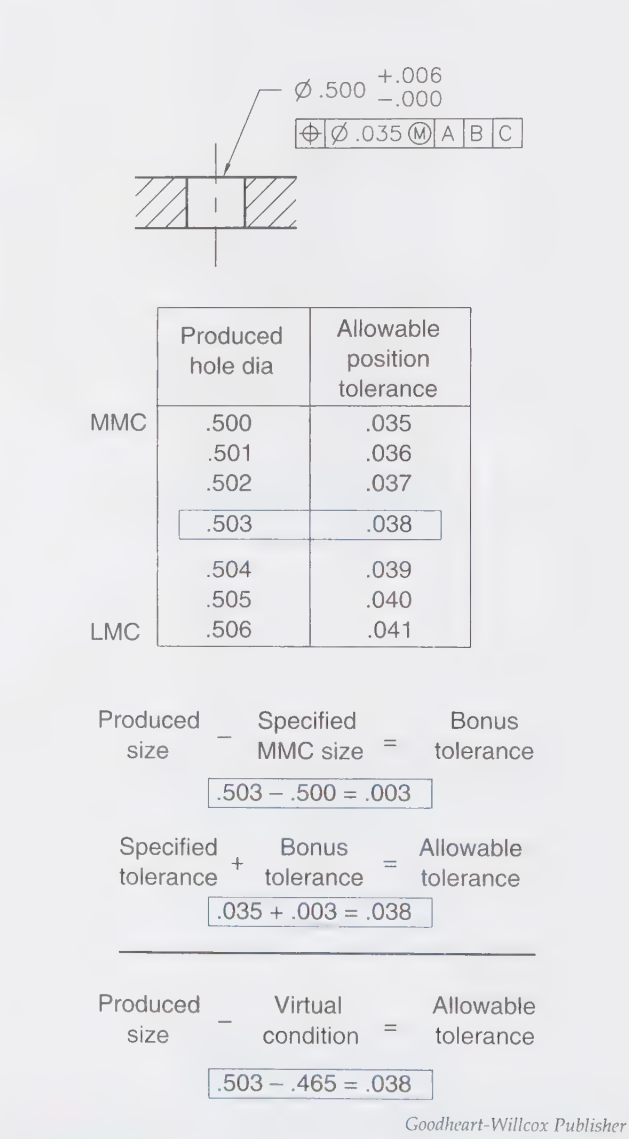
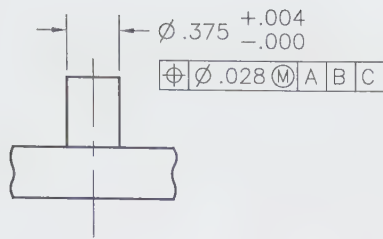


Figure 8-17. The MMC modifier, when used on the position tolerance for a hole, can significantly increase the allowable tolerance on the produced holes.



	Produced pin dia	Allowable position tolerance
MMC	.379	.028
	.378	.029
	.377	.030
	.376	.031
LMC	.375	.032

$$\text{Specified MMC size} - \text{Produced size} = \text{Bonus tolerance}$$

$$.379 - .376 = .003$$

$$\text{Specified tolerance} + \text{Bonus tolerance} = \text{Allowable tolerance}$$

$$.028 + .003 = .031$$

$$\text{Virtual condition} - \text{Produced size} = \text{Allowable tolerance}$$

$$.407 - .376 = .031$$

Goodheart-Willcox Publisher

Figure 8-18. The MMC modifier, when used on the position tolerance for a pin or shaft, can significantly increase the allowable tolerance on the produced parts.

.028" diameter. There is an allowable increase in the position tolerance if the pin departs from the MMC size. The given chart shows that for every .001" decrease in the pin diameter, there is a .001" increase in the allowable position tolerance.

The allowable position tolerance can just as easily be calculated for a pin as has already been demonstrated for a hole. The figure shows how the allowable tolerance is calculated for a pin produced at a .376" diameter. The difference between the produced size (.376") and the specified MMC size (.379") is found to determine the bonus tolerance (.003"). The bonus tolerance (.003") and the specified position tolerance (.028") are added together to determine the allowable position tolerance (.031"). Another way to calculate the allowable position tolerance for a pin is to find the difference between the virtual condition and the produced size of the feature. See [Figure 8-18](#).

Verifying Position Tolerances

Produced hole locations may be verified to meet positional tolerance requirements in any of several methods. Coordinate measuring machines may be used to determine if feature locations are within specified position tolerances. Functional gages may be used to verify that position tolerances are achieved. Coordinate measurements may also be made manually.

Coordinate measurements may be compared to the true position coordinates to determine the amount of variation that exists in each produced feature location. The difference between the true position coordinate value and the measured value is the amount of variation from true position. The variations from true position may be referred to as the delta coordinates, or delta X (ΔX), delta Y (ΔY), and delta Z (ΔZ) values. See [Figure 8-19](#).

The delta coordinates may be used to calculate the diameter of the positional variation for each hole as explained in the following paragraphs. These calculations may be performed manually or automated. Software for these calculations is commercially available. The table in Appendix A1 may be used to assist in the review of measurement data. The delta coordinates may also be plotted on grid paper to determine the amount of positional variation and this method is explained in following paragraphs.

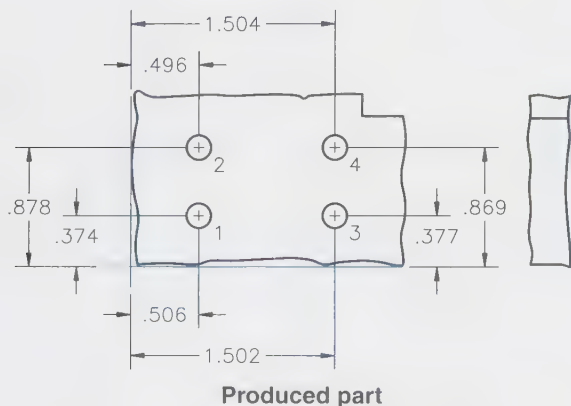
Position Calculations from Coordinate Measurements

Coordinate measurements may be used to calculate the diameter of position tolerance consumed by a measured hole. The position tolerance used, or consumed, by the produced hole location must be equal to or less than the allowable position tolerance for the hole.

To determine if a hole position is acceptable, it is necessary to obtain the measured coordinates for the produced hole location. See [Figure 8-19](#). Coordinate measurements to each of the produced hole positions are made relative to the datum reference frame. The given figure shows the measured coordinate data entered in the table. The difference between the measured coordinate data and the required coordinates defined by the drawing dimensions is found to determine the amount of X and Y variation in the location of each hole. The X and Y variations, delta coordinate values, are used to calculate the required diameter of circle to enclose the delta coordinates.

The given figure shows a drawing and a produced part. It also includes a table that shows the

The diameter of position variation resulting from the coordinate variations, in this case the X and Y coordinate variations, may be calculated using the formula in [Figure 8-20](#). After obtaining information shown for hole #1 in the previous figure, we see that the X coordinate variation is .006" and the Y coordinate variation is -.001". The calculations of the diameter position variation require squaring the two coordinate variation values (.000036" and .000001" respectively), then finding the square root of the sum of the two squares (.00608"). This value is the hypotenuse of the coordinate errors, and therefore is the radius of a circle centered on the true position and drawn through the center of the produced hole. To determine the diameter, the radius is multiplied by 2. The diameter position variation for the example hole is .0122"



Hole #	3		4	
Diameter	.191		.191	
	X	Y	X	Y
Measured location	1.502	.377	1.504	.869
Drawing dimension	1.500	.375	1.500	.875
Variation	.002	.002	.004	-.006

Goodheart-Willcox Publisher

$$\text{Diameter position variation} = 2 \sqrt{\Delta X^2 + \Delta Y^2}$$

Measured position
variation diameter

$$= 2 \sqrt{.006^2 + (-.001)^2}$$
$$= 2 \sqrt{.000036 + .000001}$$
$$= 2 \sqrt{.000037}$$
$$= 2 (.00608)$$
$$= .0122$$

Allowable tolerance	=	Specified tolerance	+	Any applicable bonus tolerance
---------------------	---	---------------------	---	--------------------------------

Produced hole size	.191	Specified tolerance	.012
MMC size	<u>-.188</u>	Bonus tolerance	<u>+.003</u>
Bonus tolerance	.003	Allowable tolerance	.015

Measured position variation must be less than or equal to allowable tolerance

$$\text{Actual position variation} \leq \text{Allowable position tolerance}$$

$$.0122 \leq .0150$$

Figure 8-20. Allowable tolerance and actual position variation may be calculated and compared to determine if a produced part is acceptable.

Because this value was determined from measured data, it may be labeled as the *measured position variation diameter* or *position tolerance consumed*. For the hole position to be acceptable, the measured position variation diameter must be less than or equal to the allowable position tolerance.

As previously explained, the allowable position tolerance is equal to the specified tolerance plus any applicable bonus tolerance. A bonus tolerance is applicable where the MMC modifier is shown in the tolerance value and the controlled feature has departed from MMC. A bonus tolerance is also possible where the tolerance includes the LMC modifier and the feature departs from LMC.

The information provided in **Figure 8-19** shows that hole #1 was produced at .191" diameter. The specified MMC is .188" diameter. The difference between the produced hole size and the specified MMC size permits a bonus tolerance of .003".

The allowable position tolerance for hole #1 is determined by adding the specified tolerance of .012" and the bonus tolerance of .003". The maximum allowable position tolerance for hole #1 at its produced size is .015" diameter. Because the measured position variation is less than the allowable position tolerance, the hole is in an acceptable location.

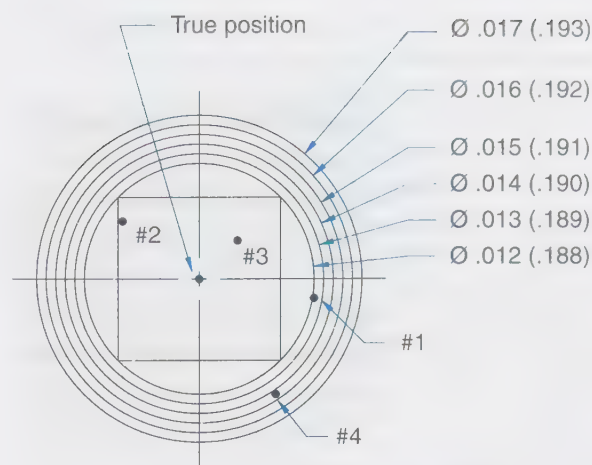
The *Coordinate-to-Diameter Conversion Table* in Appendix A1 may be used to convert coordinate values into diameter values. The values in the table are diameters.

Paper Gaging

The position tolerances for produced features may be checked for compliance using a graphical process commonly known as paper gaging. **Paper gaging** is a method of illustrating a position tolerance and the actual position of a feature in a graph to determine whether the position is within the tolerance. Depending on the accuracy needed, measurement equipment may be relatively simple or it may be high precision. Whatever measurement equipment is used, it must be able to make accurate coordinate measurements relative to the datum reference frame that is specified in the position tolerance. In addition to measuring feature locations, it is important to also measure the size of produced features.

Figure 8-21 illustrates how paper gaging may be used to verify that a hole pattern meets the required position tolerances. Data for the four holes in **Figure 8-19** is used. In that figure, the given drawing includes a position tolerance of .012" diameter at MMC, relative to datums A, B, and C.

A point is selected on the graph paper to serve as the origin, and it represents the true position for



Plotted hole location coordinates and position tolerance zones

Goodheart-Willcox Publisher

Figure 8-21. Grid paper may be used for plotting coordinate variations for a produced hole pattern. A series of concentric circles may be overlaid on the grid to determine the diameter position variation.

the measured holes. The delta X and delta Y values (variation values in the table) for the measured hole positions are plotted on the graph paper, and points are drawn to represent the centers of the holes. The grid values are assigned a large scale to make any minor plotting errors insignificant. In the given figure, each grid line represents .001".

A series of concentric circles is overlain on the graph paper with the center of the circles at the origin for the plotted points. The circles and grid are drawn to the same scale. The smallest circle in the given figure is drawn to represent a diameter of .012". Another circle is drawn at every .001" increase in diameter up to a diameter of .017".

Although the specified position tolerance is .012" diameter, the MMC modifier indicates that any hole produced larger than MMC will be allowed a bonus tolerance. The plotted location for hole #1 is at .006", -.001". The point falls slightly outside the .012" diameter circle. Because hole #1 was measured at a diameter of .191", its position tolerance is .015" diameter because of the effect of departure from MMC. Therefore, the location of hole #1 is acceptable. Plotting the location variation of all four holes and comparing the point location to the applicable tolerance zone diameter for the produced hole size shows the given part to be good.

Functional Gages

Functional gages can be produced for verification of hole positions. When many parts must be inspected, it may be less expensive to fabricate a functional gage than to measure each part

and analyze the data. Explanations regarding the usage of functional gages for position tolerances are given in the next chapter.

The preceding explanations of methods for verifying position tolerances are not intended to indicate that every fabricated part must be inspected. Process control, when appropriately applied, may minimize the number of parts that must be inspected. The previous paragraphs are intended to explain the part requirements created by position tolerances and are not intended to require a specific means for inspection.

Advantages of Position Tolerances

Position tolerances have advantages when compared to the use of directly applied coordinate tolerances for defining location requirements. One advantage is the clarity of the requirement provided by a position tolerance. When the basic dimensions, datums, and position tolerance are properly applied, there is only one correct interpretation of the requirements. The origin for coordinate tolerances is not always clear, and the accumulation of tolerances is subject to interpretation when using directly applied coordinate tolerances for the location of features. Directly applied coordinate tolerances should be avoided when specifying location requirements.

Position tolerances take advantage of the full amount of tolerance that is acceptable in the design. This is done through the application of circular zones for round features and through the allowance of bonus tolerances. Coordinate tolerances may create square or rectangular zones depending on how they are applied, and coordinate tolerances cannot take advantage of bonus tolerances.

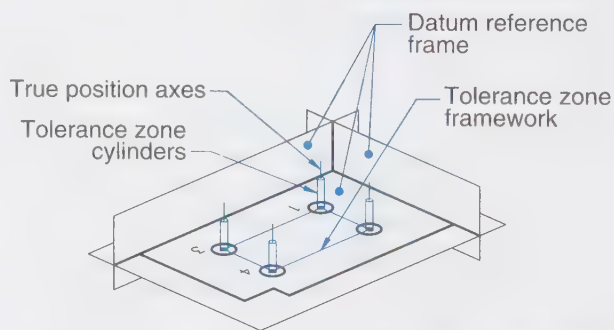
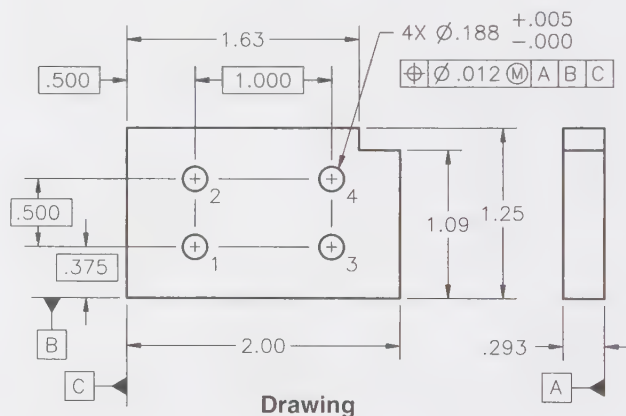
All the freedom and flexibility needed to meet any design application is available when using position tolerances. Either small or large tolerances may be specified, and they may be applied on simple or complex parts. They are applicable regardless of fabrication process and may be used on machined, cast, or molded parts as well as on sheet metal, composites, and additive processes.

Clarity of Position Tolerances

Application of position tolerances results in a clearer definition of requirements than a location tolerance defined by coordinate tolerances. There is a well-defined meaning for any properly specified position tolerance. Coordinate tolerances (plus and minus tolerances) applied on location dimensions are subject to interpretation and may result in confusion.

The clarity of a position tolerance is attributed to several factors. One significant factor is the documented meaning and interpretation provided in the ASME Y14.5-2009 dimensioning and tolerancing standard. Position tolerancing is well-defined by this standard. Written standard guidelines are important because they eliminate the need for people to assume a set of rules by which to interpret dimensions. Using a dimensioning system that has no standardized interpretation leaves a drawing subject to varying interpretations and possible disputes. There is no defined meaning of directly applied coordinate location tolerances (plus and minus tolerances) in the ASME Y14.5-2009 standard; therefore, the use of coordinate location tolerances leaves a drawing subject to varying interpretations and that potentially leaves the design requirements vague or without meaning. The use of position tolerances on features of size avoids the ambiguity of plus and minus tolerances on location dimensions.

Another contributor to the clarity of position tolerances is the fact that tolerance zones are located relative to referenced datums. See **Figure 8-22**. The datum feature references in the given figure create



Goodheart-Willcox Publisher

Figure 8-22. The position tolerance zones are located on a tolerance zone framework that includes the true positions for all features and is related to referenced datums.

a datum reference frame made of three mutually perpendicular planes. The datum reference frame provides a clear reference from which to manufacture and inspect the part. The datum reference frame created by datum feature references in a position tolerance is not affected by surface variations. Avoidance of the surface variation effects on position is a benefit of positional tolerances and is not achievable with the old system of coordinate location tolerances.

Clarity of the position tolerance requirement is further enhanced by the usage of basic dimensions to define the nominal feature locations. In **Figure 8-22**, the basic dimensions create a tolerance zone framework that is theoretically perfect and defines the true positions, which include true locations and true orientations, for the features based on the dimensions locating the holes. The tolerance zone framework is related to the datum reference frame by the dimensions shown in the drawing or the data in the solid model.

There is no tolerance accumulation on basic dimensions. Because there is no tolerance accumulation, the position tolerance zones are always centered on the true positions as located by the tolerance zone framework. This does not require that a produced part be perfect. Only the tolerance zones must be located at the true positions. The tolerated features may be anywhere within the defined tolerance zones.

Freedom exists to express the required level of control, ranging from very small to very large tolerances, when using position tolerances. One misconception about position tolerancing is that it should only be used to achieve a high degree of accuracy. Of course it may be used to express small tolerances, but it is just as useful for expressing large tolerances. The only limits on the tolerance values placed in a feature control frame are the functional requirements of the part and the available manufacturing capabilities.

Flexibility to meet a wide range of design applications is built into the position tolerancing methods. The methods are not restricted to complex parts, or limited to simple ones. Only the level of control required by the function of the part needs to be specified when using position tolerances.

Position tolerancing methods are well-defined by the current dimensioning and tolerancing standard. They are widely accepted throughout the United States and internationally. Military and defense contractors use these methods extensively. Both large and small companies recognize the benefits of this system and adopt it for utilization in their designs. It is necessary to understand and

properly utilize positional tolerancing methods to be compatible and competitive in today's industry.

Ambiguity of Coordinate Location Tolerances

Coordinate location tolerances have been used for many years, and a great number of parts have been successfully produced. The success was to some extent a result of manufacturers assuming requirements that were not specified. The past successful fabrication of poorly defined parts is not an adequate reason for continued use of a system that is ambiguous.

There currently is no standard that defines the meaning of coordinate location tolerances. A simple plate may be used to show the ambiguity of coordinate tolerances. See **Figure 8-23**.

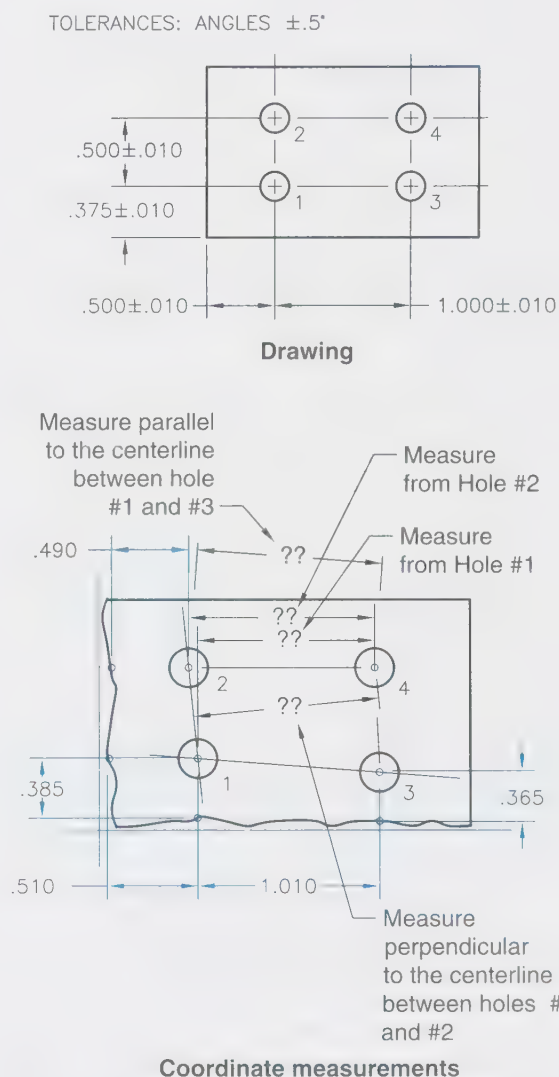


Figure 8-23. Coordinate location tolerances are ambiguous. The origin and direction for measurements are not defined. All measurements taken must be done on the basis of assumptions because no standard defines the coordinate tolerancing system.

There are four holes in the given plate. The drawing shows hole #1 dimensioned from the lower-left corner of the plate. Holes #2 and #3 are dimensioned from hole #1. It is not clear whether hole #4 is dimensioned from holes #2 and #3 or from hole #1. Edges of the plate are drawn straight and at a 90° angle to one another. Dimensions to the holes are drawn parallel and perpendicular to the edges. Coordinate tolerances of $\pm .010$ are shown on all the hole location dimensions.

One possible produced part is shown below the drawing. The edges of the part have been saw cut and are rough. The edges do not form a perfect 90° angle. This is acceptable because the shown drawing note permits a $\pm .5^\circ$ tolerance on angles.

Immediate questions start to arise as it becomes necessary to locate hole #1. Because the edges are not perpendicular, from where should measurements be made and from which edge should measurements be oriented? These are difficult questions to answer because no datums are identified or referenced.

If some assumptions are made, it is possible to make measurements for holes #1, #2, and #3. Locations for these holes could be made relative to the edges nearest the hole. Of course, surface variations will cause some location difference for the origin used to measure each of the holes. The direction of measurements is left somewhat undefined because the two edges of the part are not perpendicular. Because of the surfaces not being flat and not being perpendicular, someone may suggest creating a coordinate system and measuring from that. So now there are two options for setting the origin, and measurements will differ depending on the method selected. Someone else may make yet different assumptions and obtain different measurement results.

The given figure shows one possible position for the first three holes if we assume that measurements may be made from the imperfect part edges. Hole #1 is located .510" from the left edge and .385" from the bottom edge. Hole #2 is located .490" from the left edge and hole #3 is .365" from the bottom edge. The hole #1, #2, and #3 locations result in centerlines between them that are spread to an angle greater than 90°.

A considerable problem now exists for locating hole #4. Should measurements continue from the edges of the part, or should measurements now be made from one or more of the other three holes? If they are made from the holes, at what angles should measurements be made? With the centerlines forming an angle greater than 90°, the direction of measurement is unclear. There are no

defined answers for the questions that arise. There are no answers because there are no standards defining how the measurements are to be made.

The ambiguity in coordinate location tolerances is caused by part variations that can exist and have been shown in the given figure. Each person that attempts to determine how the part is to be measured must assume a set of guidelines. Each person that assumes different guidelines can produce parts that are different from anyone else's parts. These types of possible differences in interpretation are not acceptable if parts are expected to meet functional or assembly requirements. The confusion can be eliminated through the application of positional tolerances.

Position tolerances have not always been shown on drawings, so how is it that good parts were ever produced to coordinate location tolerances? In many situations, manufacturing processes were set up to produce good parts based on the obvious function of the part. This was done regardless of the incomplete requirements defined by the drawings.

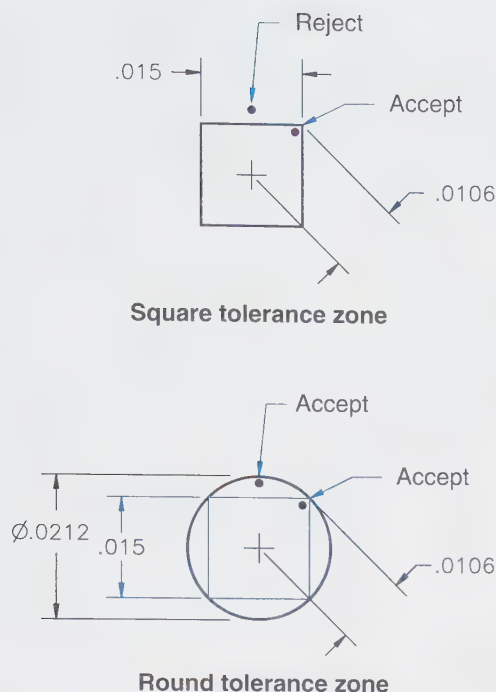
The good intentions of the manufacturing companies are not always sufficient. Not all parts produced to coordinate tolerances will work as expected. Situations have existed where parts were produced and the manufacturer believed that its parts met the drawing requirements. However, the parts would not fit into the required assembly. Whenever a manufacturer makes parts that it believes are correct to the drawing, the manufacturer wants to be paid for the parts. When a customer pays to have parts produced, the customer expects them to fit in the required assemblies. When the fabricator and the customer disagree about the requirements on a drawing, the disagreement can lead to production delays, lost profits, and broken business relationships. In extreme cases, disagreements may lead to litigation.

It is much better to completely and clearly define requirements using position tolerances than to argue about product requirements after fabrication is completed.

Increased Tolerance Zone Area

Round tolerance zones specified by positional tolerances provide a significantly increased amount of tolerance zone area when compared to the square tolerance zone created by a coordinate tolerance. See [Figure 8-24](#). The top half of the given figure shows a square tolerance zone. The square is .015" across the flats.

The crossed lines represent the true position for two holes. The X and Y coordinate variations



Goodheart-Willcox Publisher

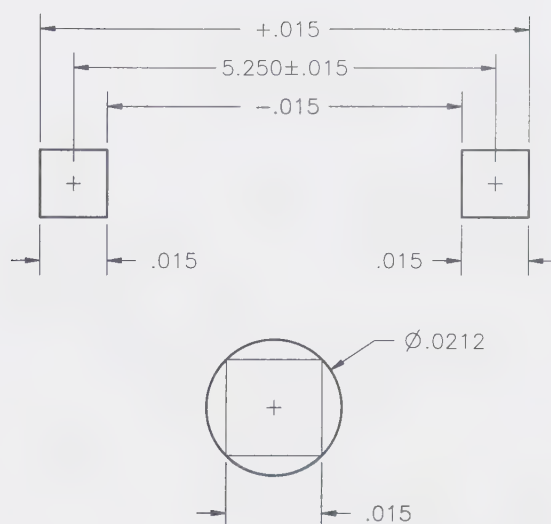
Figure 8-24. A round tolerance zone permits the same amount of variation in all directions. The square tolerance zone does not.

of two produced holes are plotted relative to the true position in the figure. Locations for the centers of the two holes are indicated by filled circles. Both of the center points are exactly the same distance (.0106") from the center of the tolerance zone. One center point is in the corner of the tolerance zone, and is therefore acceptable. The other center point is on the vertical centerline. It is located outside the square tolerance zone; therefore, it is not acceptable. The illustrated square zone forces the rejection of one of the two holes, although the magnitude of variation in the two holes is identical.

Fortunately, position tolerances may be used to avoid rejecting functionally good parts. The bottom half of the figure shows a circular tolerance zone circumscribing the square. The tolerance zone has a .0212" diameter. This size of round tolerance zone permits the same .0106" location variation in any direction; therefore, the two shown center point locations are acceptable.

Initial impressions of the improvement from a square to a circular zone may not seem significant, but calculations of the areas for the two shapes show there is actually a 57% increase in area. See **Figure 8-25**. The area of a .015" square is .000225 in². The area of a .0212" diameter circle is .000353 in².

The increased tolerance area realized with the round zones may be used to increase the producibility of parts. An increase in producibility



Goodheart-Willcox Publisher

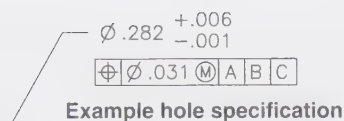
Figure 8-25. The area of a round tolerance zone is 57% greater than the area of a square zone.

can often reduce part cost. Choosing to ignore the advantages of the larger zone, and continuing to use square tolerance zones, reduces the functionally acceptable tolerance by 36%. Unnecessarily reducing the available tolerance can cause rejection or rework of functionally good parts. It is a bad practice to specify part requirements that force the rejection or rework of functionally acceptable parts.

Bonus Tolerances (Additional Tolerances)

It has been shown that using a circular tolerance zone increases the tolerance zone area by 57% over the area of a square zone. The same tolerance zone may be further increased through bonus tolerances when the MMC or LMC modifier is applied to the position tolerance.

Figure 8-26 shows how the bonus tolerance may be used to increase the acceptable amount of location variation. An example hole specification is given. Just below it, a table compiling inspection data from two holes on a produced part is shown. The resulting position variation is tabulated in the right column. The methods for calculating the diameter of the position variation were previously explained in this chapter.



		Specified location	Produced location	Position variation	
Hole #1	X	1.000	1.009	+.009	Ø.0241
	Y	2.500	2.492	-.008	
Hole #2	X	1.375	1.390	+.015	Ø.0316
	Y	2.500	2.505	+.005	

Tabulated hole location data

	Specified MMC	Measured diameter	Bonus position tolerance
Hole #1	.281	.284	.003
Hole #2	.281	.285	.004

Measured size – Specified MMC = Bonus position tolerance

	Allowable tolerance	Position variation	Acceptable part
Hole #1	.034	.0241	Yes
Hole #2	.035	.0316	Yes

Specified tol at MMC + Bonus = Allowable tolerance

Goodheart-Willcox Publisher

Figure 8-26. Departure from MMC results in bonus tolerance that further increases the size of the allowable position tolerance zone.

A second table shows how the allowable bonus tolerance on each of the holes is determined. The bonus tolerance on hole #1 is .003". The .003" bonus tolerance is added to the specified .031" position tolerance and results in an allowable position tolerance of .034" for hole #1. The bonus tolerance on hole #2 is .004". This value added to the specified .031" position tolerance results in an allowable position tolerance of .035" on hole #2.

The added bonus tolerance on holes #1 and #2 significantly increases the allowable variation on the part. However, the variation will not prevent the part from assembling properly.

The advantage of a bonus tolerance for location can only be achieved by using position tolerances and the MMC or LMC material condition modifiers. There is no way of taking advantage of bonus tolerances when using coordinate (plus and minus) tolerances. Coordinate tolerances provide a fixed tolerance value. It remains unchanged regardless of how large the holes are produced.

There are several ways in which bonus tolerances may be used to increase producibility. Careful planning can actually let the machine shop take advantage of the bonus before any holes are produced. For the example holes in **Figure 8-26**, the holes could purposefully be drilled larger than

MMC to increase the allowable position tolerance. If holes are drilled at a diameter of .285", the resulting bonus tolerance of .004" can be added to the specified position tolerance to obtain an allowable position tolerance of .035" diameter.

Bonus tolerance may also be used when reworking holes that are slightly outside the specified position tolerance. All that needs to be done to correct the out-of-tolerance condition is to open the hole diameter within size limits and without drifting (pulling) the hole further off center. This increased hole size will result in bonus tolerance that may be used to accept the produced hole location.

Using the hole specification in **Figure 8-26**, consider a part in which one hole is produced too far out-of-position. Consider a hole that has a .281" diameter and a position of .032" diameter. For this hole, a bonus of only .001" diameter is needed. This means the hole only needs to be increased to a diameter of .282".

Special Applications

Position tolerances may be applied to features of size other than simple clearance holes. Some common applications include counterbored holes, threaded holes, and noncylindrical features of size. Some application requirements affect how the tolerance is specified and where the tolerance zone is located. Although the default location of a tolerance zone is along the length of the controlled feature, the zone is sometimes specified to lie outside the feature. This is done with a projected tolerance zone.

Position tolerances must include datum feature references to establish requirements for the position between the toleranced features and other features on the part. There are some special applications when the toleranced features are not related to datums, but in those cases the toleranced features must serve as the primary datum feature for location tolerances applied to other features on the part. This is explained in a later section of this text.

Counterbored Holes

A common use for a counterbored hole is to permit the installation of a flush-head screw or bolt. It is only possible to install the screw or bolt if the clearance hole and the counterbore are the correct size and located with sufficient accuracy. Counterbores are often produced in a separate machining operation from the through hole, so the two features may not lie on the same axis.

Positional tolerance specifications may be applied that will ensure assembly of the screw or

bolt into the counterbored hole. There are three methods of achieving the desired results. See **Figure 8-27**. The first method uses a single position tolerance specification next to the hole and counterbore callout. The position tolerance applies to both features. The two features act independently and may float in opposite directions within the specified zone.

If different diameter tolerance zones are desired for the two features, then a position tolerance specification may be placed by the hole callout and another by the counterbore callout. The features act independently, and may travel to opposite sides of their respective tolerance zone.

It is possible to reference the counterbore to the hole if, for some reason, there is a need to do this. In this case, a position tolerance is applied to the clearance hole. If there are four holes in the hole pattern, then the position tolerance applies to all four holes. A datum feature symbol is applied to one of the holes, with a note stating 4X INDIVIDUALLY. This indicates that each hole is going to serve as a datum for its counterbore. Omission of the notation would indicate that all four holes act simultaneously to establish a datum.

A position tolerance is applied to one of the counterbores to show the counterbore position relative to the clearance hole. The note of 4X INDIVIDUALLY is shown adjacent to the position tolerance for the counterbore. This allows each of the counterbores to act as a single entity with its position tolerance located relative to the axis of its clearance hole.

Counterbore diameter calculations are completed using the same formulas as are used for calculating the clearance hole diameter. The maximum material condition for the screw head is used in calculations to determine the counterbore diameter. If there is a fillet radius at the bottom of the counterbore, the diameter of the flat at the bottom of the counterbore should be used in the calculations to ensure the fastener head does not ride up on the fillet.

Application to Threads

Position tolerances applied to a threaded feature are assumed to be applicable to the location of the pitch cylinder axis. This is because threads are self-centering on the pitch cylinder.

The pitch diameter, therefore, serves as the logical controlling feature. The feature control frame may be placed adjacent to the thread specification, or it may be connected to the hole with a leader. See **Figure 8-28**. When a position tolerance is applied to a thread, the axis method of

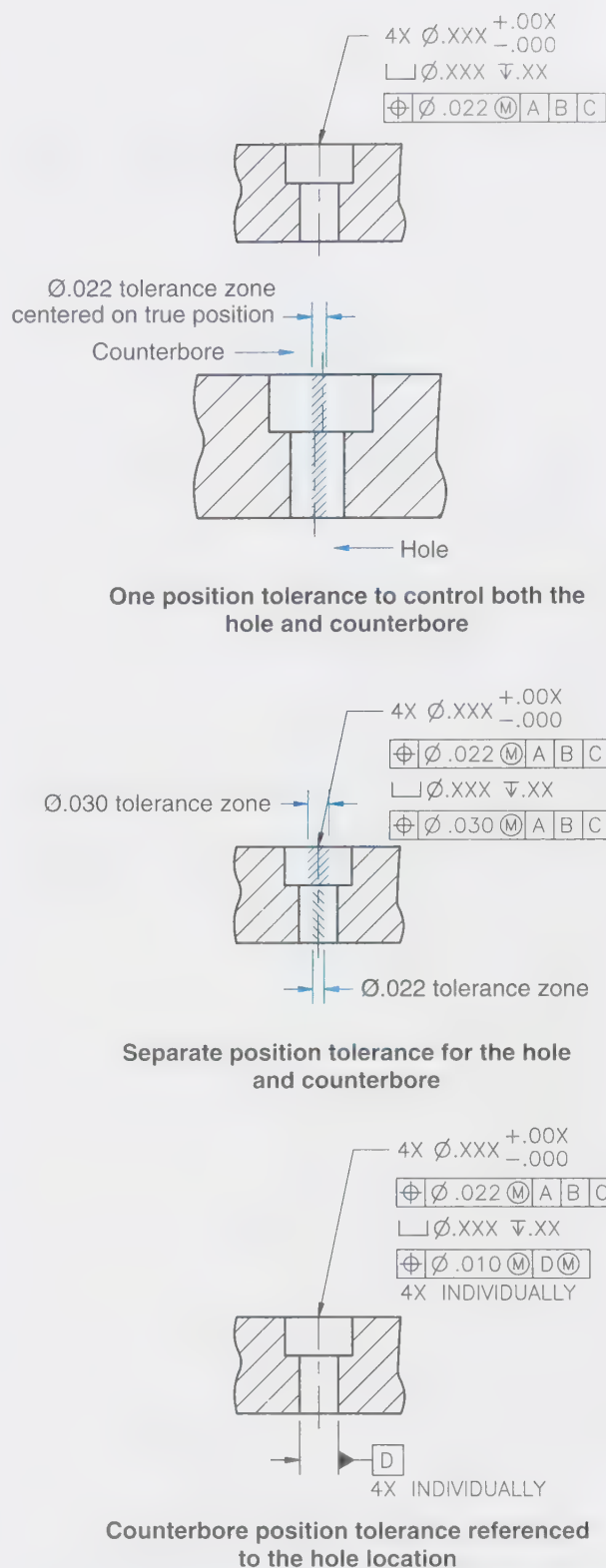
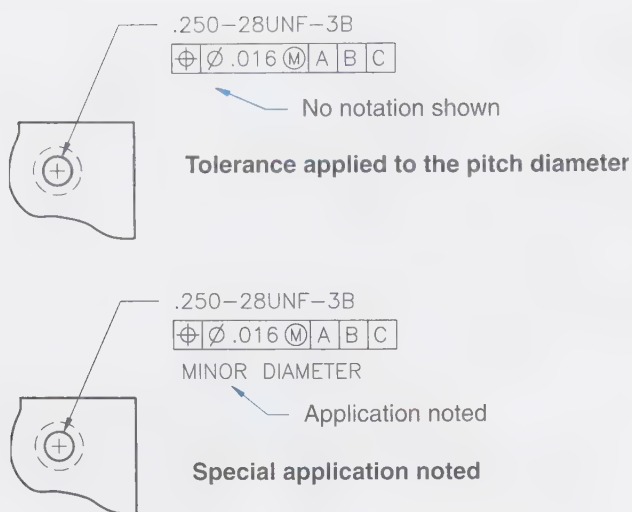


Figure 8-27. Three methods for tolerancing the location of a counterbore are shown.

verification must be used. The surface method of verification is not possible because the pitch cylinder is a derived feature and not a surface.



Goodheart-Willcox Publisher

Figure 8-28. A position tolerance applied to a thread is understood to control the location of the pitch diameter unless noted otherwise.

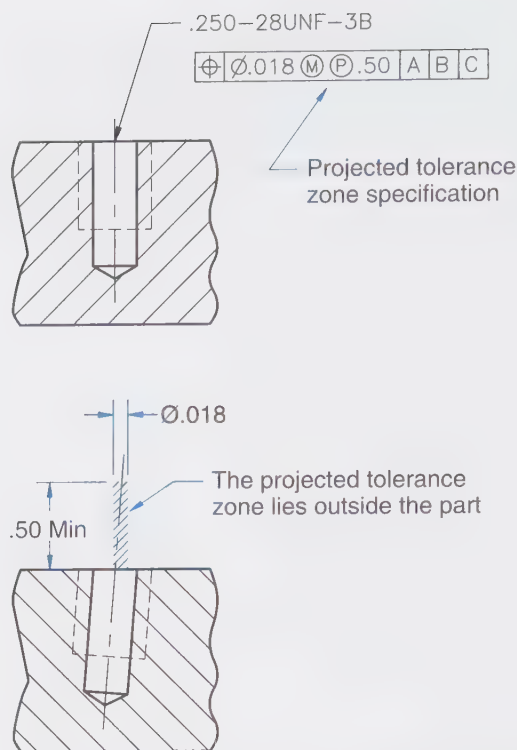
If it is necessary to apply a position tolerance to control either the minor or major diameter location of a threaded feature, then a notation must be placed under the feature control frame. The notation should simply state MINOR DIAMETER or MAJOR DIAMETER. Control of these features in place of the pitch diameter should be approached with caution because there is no certainty that the various diameters of a thread will actually be perfectly coaxial when the thread is produced.

Projected Tolerance Zones

Position tolerances applied to threaded holes, and other fixed features, often include a projected tolerance zone. See **Figure 8-29**. A *projected tolerance zone* lies outside the toleranced feature. This is appropriate for a feature such as a threaded hole, because the thread actually locates a screw that extends outside the hole. The location of the screw where it passes through the adjacent part is what is important, so the tolerance zone should extend a distance at least equal to the mating part thickness.

One method for indicating a projected tolerance zone is to show the circled "P" symbol and a projection distance following the position tolerance value and any shown material modifier. The given example shows a requirement for a projected zone to extend .50" outside the part.

Verification of a projected position tolerance is typically achieved using the axis method. The surface method of verification is difficult to achieve and may not be sufficiently accurate. In the case of a projected position tolerance on a threaded hole, the surface method is not feasible.

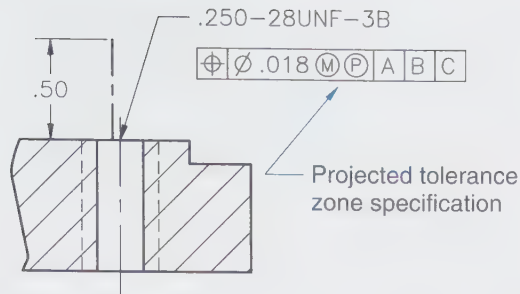


Goodheart-Willcox Publisher

Figure 8-29. Projected tolerance zones may be specified with the projection distance shown in the feature control frame when the direction of the projection is understood.

The projected tolerance zone lies outside of the feature. On the shown part, the axis of the thread must be within the specified .018" diameter tolerance zone. The tolerance zone extends from the part surface to a distance .50" outside the surface. There is no requirement defined for the thread location on the inside of the part. To position tolerance the thread location inside the part, another position tolerance may be applied without the projected tolerance zone symbol.

A projected tolerance zone may also be indicated in an orthographic view with only the circled "P" inside the feature control frame and a dimension applied directly to the controlled feature. See **Figure 8-30**. A heavy chain line is drawn adjacent to the controlled axis and a dimension applied to it. This is especially useful in orthographic views when the controlled feature is a through hole. The dimension and chain line clearly indicate to which side of the part the zone is to project. This method of showing direction in orthographic views is not recommended for solid models. When a projected tolerance is specified on a solid model, the position tolerance leader is applied on the side of the part from which the tolerance projects.



Goodheart-Willcox Publisher

Figure 8-30. Projected tolerance zones may be specified with the projection distance dimensioned on the feature.

Bidirectional Position Tolerance

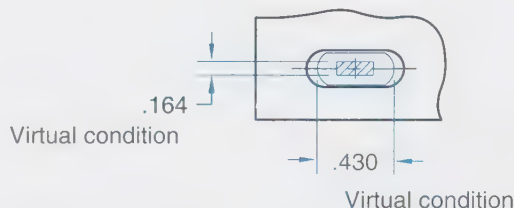
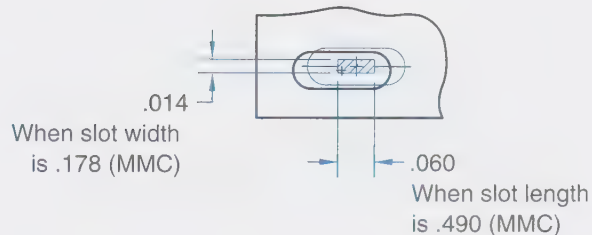
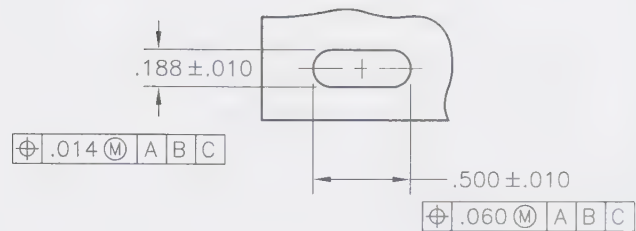
It is sometimes desirable to permit more position tolerance in one direction than the other. This is referred to as **bidirectional position tolerance**.

A common application of bidirectional position tolerance is on a slotted hole. See **Figure 8-31**. Two feature control frames are used to show the amount of tolerance on the hole, with each of the feature control frames specifying allowable variation along one axis. The given figure shows a position tolerance of .014" applied to the .188" slot width dimension. A position tolerance of .060" is applied to the .500" slot length dimension. Neither of the two position tolerances includes a diameter symbol. When applied in a solid model, the feature control frames are placed in the same plane as the two size dimensions.

The tolerance zone created by the bidirectional specification is rectangular. This tolerance zone cannot take full advantage of the functionally acceptable zone, but it does provide advantages over coordinate tolerances. The bidirectional zone is specified relative to a datum reference frame. This gives a well-defined true position. The figure shows an interpretation with the tolerance zone included. An acceptable location of an MMC slot is shown with its center within the allowed tolerance zone.

The bidirectional zone may be specified with the MMC or LMC modifier. This provides the advantage of having bonus tolerances. The given figure includes the MMC modifier on each tolerance. This allows the rectangular zone to increase in size as the slot departs from the MMC.

With the tolerances specified MMC, a virtual condition is created, and the surface of the slot is not permitted to violate the virtual condition boundary. There are actually two virtual conditions, one for the length and one for the width of the slot. The slot must meet the size requirements and both position tolerance requirements.



Goodheart-Willcox Publisher

Figure 8-31. Two feature control frames are used to show a bidirectional position tolerance.

Noncylindrical Features of Size

Location requirements for slots, tabs, rails, and other noncylindrical features of size may be specified using position tolerances. See **Figure 8-32**. The tolerance specification is placed adjacent to the size dimension of the feature to be controlled, or it may be attached to the dimension line. The feature control frame must not be placed on one of the extension lines. Placement on an extension line would incorrectly indicate a tolerance on one surface, and position tolerances are not applied to surfaces.

The first example in the given figure shows a position tolerance of .016" on a slot. The tolerance zone is referenced to datum feature A at RMB, which is the width of the part. Datum plane A is established at the center of the actual mating envelope that contacts the two sides of the part. The unrelated actual mating envelope is used in this application since it is the primary datum. Because the controlled slot is centered on the referenced datum, the slot is understood to be symmetrically located. For this reason, no dimension is needed to locate the slot.

The tolerance zone for the slot center plane is a rectangular zone .016" wide. The tolerance zone is centered on datum plane A. The tolerance zone

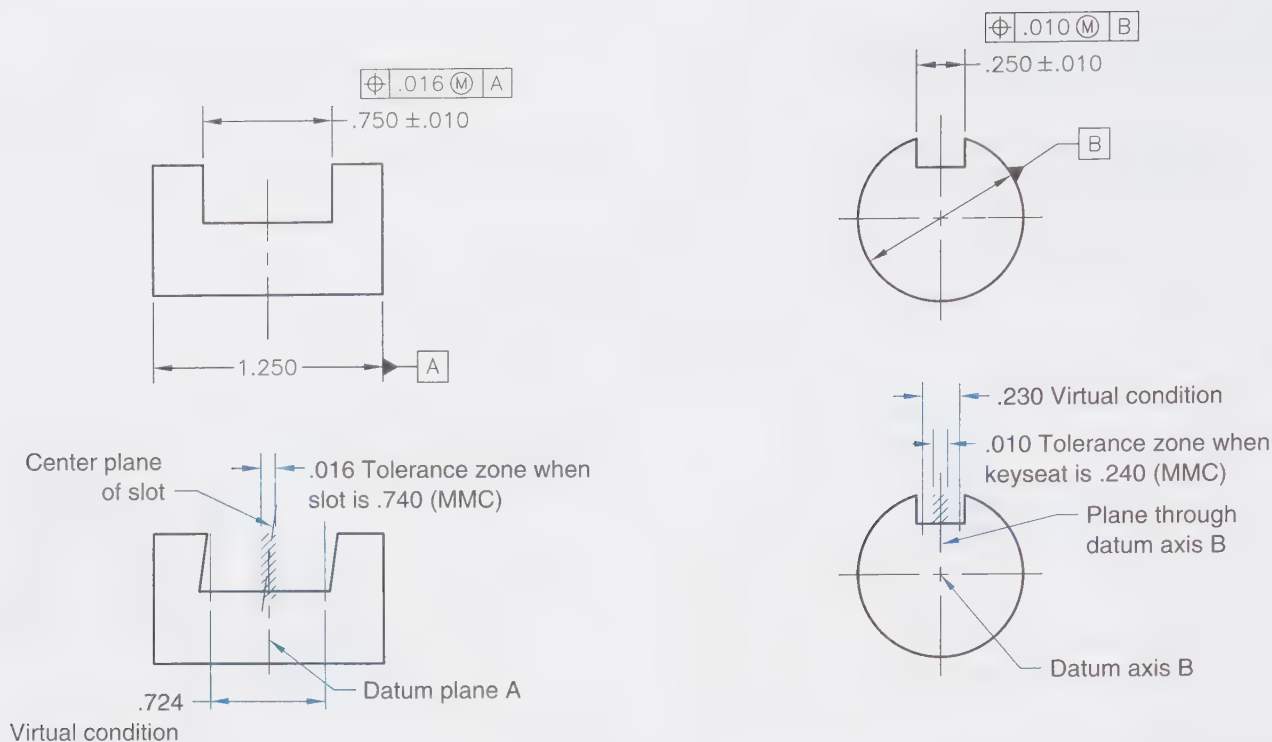


Figure 8-32. A position tolerance applied to a rectangular feature of size creates a rectangular tolerance zone that controls the location of the feature's center plane.

has a height equal to the slot, and extends the full length of the slot. The center plane of the slot must fall inside the tolerance zone. Sides of the slot may be in any condition provided the limits of size are met, the center plane falls inside the position tolerance zone, and the sides of the slot do not violate the virtual condition boundary. For a slot, the virtual condition is equal to the MMC minus the position tolerance.

The second example shows a keyseat in a shaft. A position tolerance of .010" is applied to the width of the keyseat. It is referenced to datum axis B at RMB. Datum axis B is established by the center of the actual mating envelope that contacts the outside diameter of the shaft. The .010" tolerance zone is centered on the datum axis.

The position tolerances are placed adjacent to the slot width dimensions, and therefore only define a tolerance for the position of the slot width. Neither of the position tolerances applied to the shown slots has any effect on the bottoms of the slots. The depth of the slot is controlled by either a directly toleranced dimension or a profile tolerance.

Bounded Feature

A position tolerance may be applied to a bounded feature that has a profile tolerance

applied to it. The profile tolerance establishes a maximum material boundary, and the position tolerance is used to define the allowable location variation for the maximum material boundary. See [Figure 8-33](#). Profile tolerances are explained in another chapter. The position tolerance value shown in the feature control frame is a total width tolerance. That means that the .010" position tolerance in the feature control frame creates a virtual condition boundary that is .005" wide on all sides of the boundary. The .005" wide zone on each side results in a total width .010" tolerance when opposite sides are considered. Applying position tolerances in combination with profile tolerances is explained in more depth in Chapter 11.

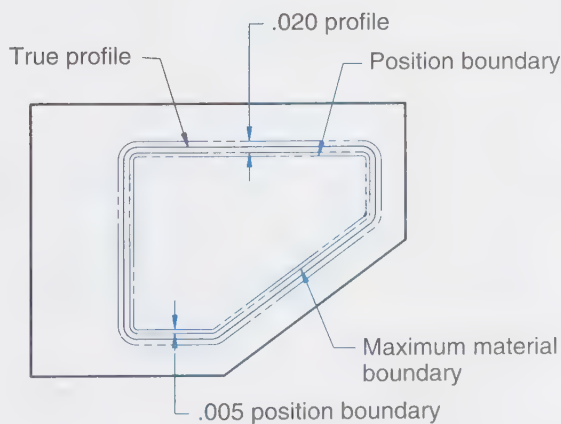
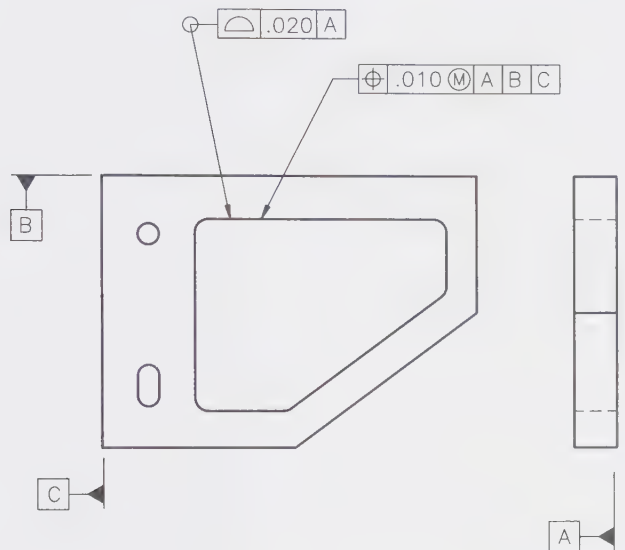
Previous standards required the notation BOUNDARY beneath the position tolerance feature control frame. The notation is no longer required.

Position Tolerance with No Datum Feature Reference

Datum feature references are required on most positional tolerances, but there are exceptions, as has been briefly explained in previous paragraphs. See [Figure 8-34](#). The given part has three cylinders that are 2.000" diameter and they lie on a common axis. A position tolerance of .002"

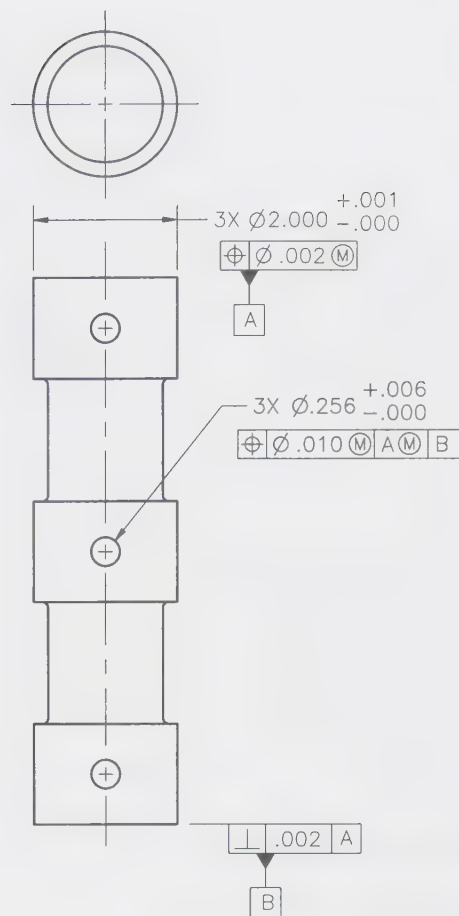
diameter is applied to define the amount of position variation that is permitted between the three features. There is no datum feature reference in the position tolerance feature control frame that is applied to these three cylinders.

The specified tolerance establishes a tolerance zone framework that is an axis through the three cylinders. It has no orientation or location requirement relative to any other feature. This practice is only permitted on this example because the tolerated features are identified as datum feature A and are referenced as the primary datum feature in other feature control frames.



Goodheart-Willcox Publisher

Figure 8-33. A position tolerance may be applied to define the position tolerance of a bounded feature when a maximum material boundary is created with a profile tolerance.



Goodheart-Willcox Publisher

Figure 8-34. A position tolerance applied to coaxial cylinders may not include any datum references if the feature serves as a primary datum feature.

Chapter Summary

- ✓ Position tolerances are used to specify the required location accuracy for features of size.
- ✓ Datum feature references are typically required on position tolerance specifications.
- ✓ RFS is assumed on position tolerances and RMB is assumed on any datum feature references.
- ✓ The position tolerance zone for a hole is typically cylindrical in shape.
- ✓ True positions for holes are defined with basic dimensions when position tolerances are applied to the holes.
- ✓ The MMC and LMC modifiers may be used to permit tolerance zones to increase, with the amount of increase dependent on the size

of the unrelated actual mating envelope of the toleranced feature for MMC and the size of the unrelated actual minimum material envelope for LMC.

- ✓ Two simple formulas permit the calculation of position tolerances for fixed and floating fastener conditions.
- ✓ Bonus tolerances are a result of size variation from the MMC or LMC when the MMC or LMC modifier is applied. Bonus tolerances permit significant increase in the tolerance zone size, depending on the actual produced size of the controlled feature.
- ✓ Inspection data showing coordinate measurements from produced parts may be used to calculate the position variation or plotted on grid paper to determine if position tolerance requirements have been met.
- ✓ Position tolerances are not limited in application to simple through holes. They may be applied to counterbored holes, threaded holes, blind bottom holes, noncylindrical features of size, and bounded features.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. One position tolerance specification may be used to control ____ at a time.
 - A. one feature
 - B. one or more features
 - C. two or more features
 - D. None of the above.
2. A feature control frame that specifies a position tolerance must include ____.
 - A. a tolerance value
 - B. a diameter symbol
 - C. implied datums
 - D. All of the above.
3. Beginning in ____, it was no longer permitted to use implied datums and mix directly applied tolerances with basic dimensions.
 - A. 1989
 - B. 1982
 - C. 1973
 - D. 1966
4. An RFS material condition modifier is assumed on ____ unless indicated otherwise.
 - A. position tolerance values
 - B. coordinate tolerances
 - C. size tolerances
 - D. none of the tolerances
5. The best position tolerance zone shape to maximize the available tolerance for a round hole is ____.
 - A. square
 - B. rectangular
 - C. cylindrical
 - D. spherical
6. Datum feature references in a position tolerance specification result in measurements being made from ____.
 - A. datum feature surfaces
 - B. assumed locations
 - C. reference points
 - D. a datum reference frame
7. The tolerance zone is unchanged by a produced feature size when ____ is applicable.
 - A. LMC
 - B. RFS
 - C. MMC
 - D. All of the above.
8. A tolerance specification on a hole has an MMC modifier. For every .002" departure of the hole diameter from MMC size, the tolerance zone increases ____.
 - A. .001"
 - B. .002"
 - C. .004"
 - D. None of the above.
9. A dowel pin pressed into one part and passing through a clearance hole in another part is an example of a(n) ____ fastener application.
 - A. fixed
 - B. floating
 - C. standard
 - D. unusual
10. If a hole is .001" out of the acceptable tolerance zone and the tolerance is specified at MMC, it might be possible to make the hole acceptable by ____.
 - A. reducing the hole diameter
 - B. moving the hole
 - C. increasing the hole diameter
 - D. None of the above.

11. A round tolerance zone has _____% more area than a square zone that is inside the round zone.
A. 100
B. 57
C. 25
D. None of the above.
12. The axis of a counterbore _____ the axis of the hole in their respective position tolerance zones.
A. is free to move in a direction opposite of
B. must move in the same direction as
C. must remain coaxial with
D. None of the above.
13. A position tolerance applied to a thread controls the location of the _____ diameter unless indicated otherwise.
A. minor
B. pitch
C. major
D. root
14. A position tolerance applied RFS to a slot controls _____ of the slot.
A. the center plane
B. the axis
C. one side
D. both sides
15. An advantage of using position tolerances is _____.
A. clarity
B. flexibility in usage
C. MMC bonus tolerance
D. All of the above.

True/False

16. *True or False?* Position tolerances are used to specify the required location accuracy for features of size.
17. *True or False?* Implied datums are used in design documentation to allow the machinist to select the features to use as datums.
18. *True or False?* A floating fastener condition exists when a bolt passes through a clearance hole in one part and threads into a hole in the second part.
19. *True or False?* A round tolerance zone has a smaller area than a square tolerance zone for the same application.
20. *True or False?* Bonus tolerances cannot be achieved when using coordinate tolerances.
21. *True or False?* If one position tolerance is specified for a hole and counterbore, then the counterbore and the hole have the same size tolerance zone.

Fill in the Blank

22. _____ dimensions must be used to define the true positions for features that have position tolerances applied.
23. The _____ and _____ modifiers permit the tolerance zone size to change in relationship to the produced feature size.
24. Rule # _____ of the current standard requires that RFS be assumed as applicable on position tolerances when no modifier is shown.
25. A position tolerance zone for a hole is centered on the _____ position of the hole.
26. A(n) _____ fastener condition exists when a bolt passes through clearance holes in two parts.
27. A(n) _____ tolerance zone extends outside the feature that is controlled.
28. A position tolerance applied to a noncylindrical feature of size should not include a(n) _____ in the feature control frame.

Short Answer

29. Create a feature control frame to require position within a diameter of .026" at MMC relative to datum feature A primary, B secondary, and C tertiary.
30. Create a feature control frame to require position within a diameter of .018" at RFS relative to datum feature B primary, E secondary, and F tertiary.
31. Explain why implied datums are no longer permitted.
32. Explain what is meant by *true position*.
33. Show the formula used to calculate a position tolerance for a fixed fastener condition.
34. Show the formula used to calculate a position tolerance for a floating fastener condition.
35. Sketch the symbol for a projected tolerance zone.

Application Problems

Each of the following problems requires calculations related to position tolerances. All of the problems may be solved using the information contained in this chapter.

36. A hole specification and position tolerance are given. What is the allowable position tolerance for a hole produced at a .502" diameter?

$$\begin{matrix} \varnothing .500 & +.006 \\ & -.000 \end{matrix}$$

\oplus	$\varnothing .018$	(M)	A	B	C
----------	--------------------	-----	---	---	---

37. A hole specification and position tolerance are given. What is the allowable position tolerance for a hole produced at a .384" diameter?

$$\begin{matrix} \varnothing .381 & +.005 \\ & -.001 \end{matrix}$$

\oplus	$\varnothing .036$	(M)	A	B	C
----------	--------------------	-----	---	---	---

38. Calculate the position tolerance to be applied to the holes for two parts in a floating fastener application. Use the given information.

Bolt MMC: .250"
Clearance Hole MMC: .281"

39. Calculate the position tolerance to be applied to the holes for two parts in a floating fastener application. Use the given information.

Bolt MMC: .190"
Clearance Hole MMC: .221"

40. Calculate the position tolerance to be applied to the holes for two parts in a fixed fastener application. Use the given information.

Bolt MMC: .164"
Clearance Hole MMC: .188"

41. Calculate the position tolerance to be applied to the holes for two parts in a fixed fastener application. Use the given information.

Bolt MMC: .190"
Clearance Hole MMC: .220"

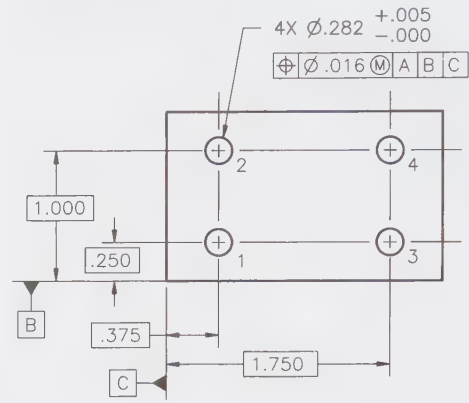
42. Calculate the required MMC clearance hole size for the given floating fastener application.

Bolt MMC: .250"
Position Tolerance: .016"

43. Calculate the required MMC clearance hole size for the given fixed fastener application.

Bolt MMC: .312"
Position Tolerance: .018"

44. A hole is dimensioned at X = 1.500" and Y = 1.250". It is produced at X = 1.505" and Y = 1.246". Calculate the diameter of the position variation for the produced hole.
45. A drawing of a simple plate with four holes is given. Inspection data showing produced hole sizes and locations is also provided. The data has already been analyzed to determine delta X and delta Y variation. Calculate the diameter position variation for each of the four holes.



Hole	DIA	Location		Variation		Diameter position variation
		X	Y	Δ X	Δ Y	
1	.284	.378	.242	.003	-.008	
2	.285	.379	1.002	.004	.002	
3	.284	1.754	.248	.004	-.002	
4	.285	1.755	1.003	.005	.003	

46. Use the inspection data from problem 45 to answer this question. Determine the amount of bonus tolerance for each of the four holes in the produced part. Using the bonus tolerance, calculate the total allowable position tolerance for each hole. Use the data from your answer to complete the given table.

Hole number	Bonus tolerance	Allowable position tolerance
1		
2		
3		
4		

Chapter 9

Position Tolerancing— Expanded Principles, Symmetry, and Concentricity

Objectives

Information in this chapter will enable you to:

- ▼ Explain functional gaging methods for checking hole position tolerances specified at MMC.
- ▼ Specify and explain composite position tolerance specifications.
- ▼ Explain the effect of using identical datum feature references in multiple position tolerance specifications.
- ▼ Specify separate pattern requirements for groups of features not acting as a single pattern.
- ▼ Specify position tolerances for in-line holes.
- ▼ Specify tolerances to control symmetry.
- ▼ Control coaxial features with position or concentricity tolerances, depending on the given application.

Technical Terms

actual mating envelope
concentricity
derived median line
derived median plane
feature-relating tolerance zone framework (FRTZF)
pattern-locating tolerance zone framework (PLTZF)
simultaneous requirement
symmetry
unrelated actual mating envelope

Introduction

Location tolerance applications include position, symmetry, and concentricity. Position tolerancing methods permit a great deal of flexibility

in the level of control that is specified on a drawing or in a solid model. The single-segment position tolerance specification presented in the previous chapter is adequate for many situations, but it falls short of meeting all the levels of control necessary to meet complex design requirements. The availability of composite position tolerances and multiple single-segment feature control frames provides a significant increase in the flexibility of what may be controlled.

A special functional application of location tolerances is to control symmetry. Depending on the application, symmetry is accomplished with a position tolerance that is applicable RFS, MMC, or LMC. There is a symmetry tolerance symbol that has a very complex meaning, and it may be used in the extremely rare situations where it is needed. A symmetry tolerance applied RFS is different than a position tolerance applied RFS.

The practice of using a symmetry symbol was not included in the 1982 standard, but was returned in the 1994 standard and made acceptable for certain RFS applications. Its usage is briefly explained in this chapter, although it is expected that this symbol will seldom be needed.

The allowable variation between features that lie on a common axis is usually controlled by position, profile, or runout tolerances. There is a concentricity symbol that may be used on coaxial features, but its complex meaning is not intuitively what the term may seem to indicate. Concentricity is briefly explained in this chapter, although it is expected this symbol will seldom be needed.

Relatively simple means of calculating position variation and paper gaging to determine the position variation relative to the specified tolerances were presented in the last chapter. Those methods require data to be gathered and analyzed.

Although paper gaging is a valuable process for many situations, gathering and analyzing data can be a significant expense if it is necessary to inspect a large quantity of parts.

Functional gages as shown in **Figure 9-1** provide another means of verifying that features are within specified position tolerances, and this chapter shows how they may be used. This chapter does not provide direction regarding how to calculate gage tolerances. Calculation of gage tolerances is an extensive subject and various methods are used depending on the inspection philosophy that is applicable. Any dimensions shown on functional gages in the figures of this textbook are intended to reflect the theoretically exact boundary that is to be verified on the part.

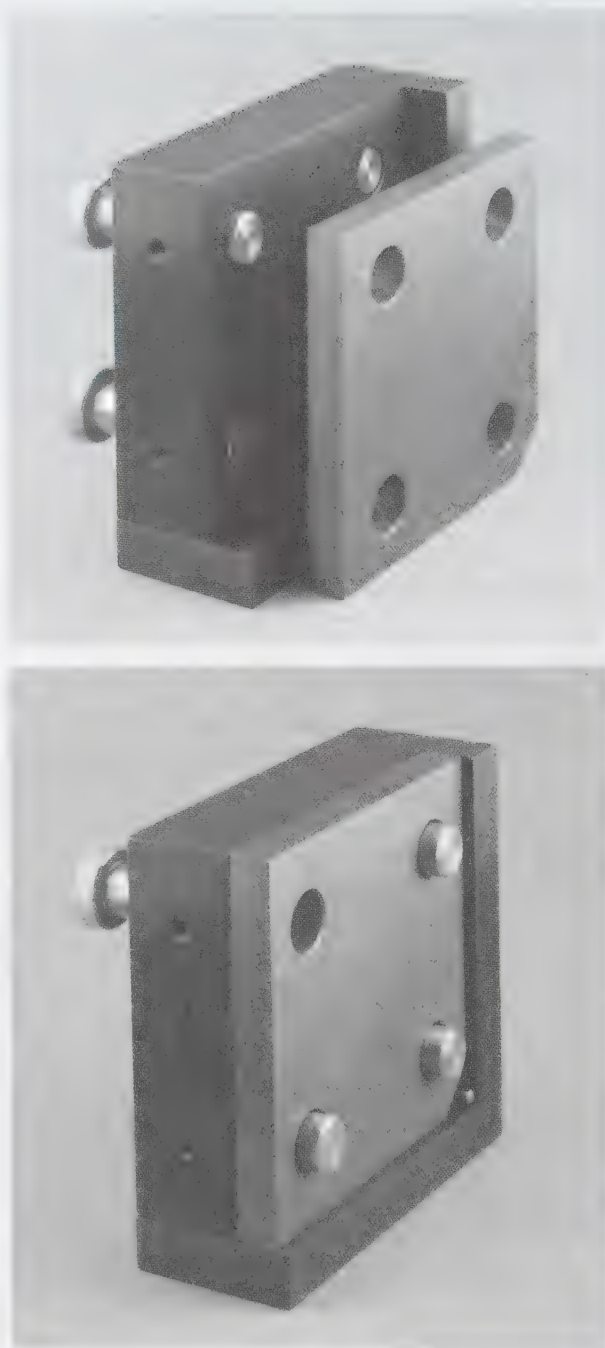
Position tolerancing methods are intended to provide a method of specifying tolerances that reflect the functional requirements of the parts. Expressing functional requirements through properly applied tolerances not only maximizes the available tolerance and improves the producibility of the parts, but it can also make it possible to check the parts with functional gages or electronic means that in effect gage the parts through analytical processes.

Although composite tolerance specifications were included prior to the 1994 standard, complex applications were not explained in detail. This resulted in varying interpretations of composite tolerances. To minimize the potential for variation in interpretation, an extensive effort was made by the ASME Y14.5 committee to expand and clarify the correct meaning. Both the 1994 and current standards expanded the explanations to provide clarity.

During the extensive development efforts related to this subject within the dimensioning and tolerancing standard, many concepts were investigated. The direction set by the 1994 standard has carried forward into the current standard and been advanced. The concepts explained in this text are consistent with the concepts defined by the standard.

Composite Position Tolerances

A single-segment (single-line) position tolerance specification creates tolerance zones that are in fixed locations relative to a datum reference frame. This results in one tolerance zone for each controlled feature. The tolerance zone on each feature has the same effect on the true position distance between features as it has on the distance of the features from the datum reference frame.



Goodheart-Willcox Publisher

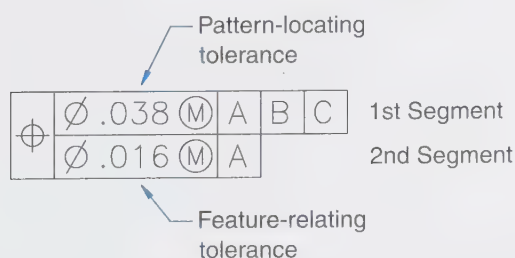
Figure 9-1. The upper photograph shows a functional gage and a part adjacent to it. The lower photograph shows the part in the functional gage with three of the four pins extended through the holes to confirm position tolerances for those holes.

Consider a single-segment position tolerance of .010" applied to two holes and referenced to datum reference frame A, B, and C. Tolerance zones measuring .010" diameter are located at true positions relative to one another and to the datum reference frame. The tolerance zones are not permitted to move relative to each other or relative to the datum reference frame.

The single-segment position tolerance creates the same level of control on the location of a hole pattern as it does for the location of holes within a pattern. There are many applications for which this level of control is acceptable and appropriate. There are other applications for which the location of holes relative to the edge of a part is less critical than the location of holes within the hole pattern.

In many design applications, there is more importance placed in the alignment of holes on mating parts than there is in the alignment of edges on those parts. This situation calls for a means of specifying two levels of control on a hole pattern. A composite position tolerance is used to specify the two needed levels of control. One level of control is defined by what is known as a *pattern-locating tolerance*. The pattern-locating tolerance allows a relatively large tolerance relative to the referenced datums, which are often established by the edges of the part. The second level of control is defined by what is known as a *feature-relating tolerance*. This feature-relating tolerance defines a relatively small tolerance to control locations of holes relative to one another.

A composite position tolerance specification has a distinct appearance that makes it easy to recognize. See **Figure 9-2**. The shown feature control frame has two segments (lines) but only one position tolerance symbol. The requirement to use a single tolerance characteristic symbol is clearly defined in the standard. The single tolerance symbol is the indication that the feature control frame is a composite tolerance. The feature control frame for a composite position tolerance shall not have two tolerance characteristic symbols. Showing two position tolerance symbols is a mistake, because two symbols indicate two single-segment feature control frames. That results in requirements that are different from those for a composite tolerance. The differences between a composite position tolerance and two single-segment feature control frames will be explained in this chapter.



Goodheart-Willcox Publisher

Figure 9-2. The feature control frame for a composite position tolerance includes a pattern-locating tolerance and a feature-relating tolerance.

People often incorrectly refer to each segment of a composite tolerance as a line of the tolerance. A “segment” is the correct term for each level of requirements in a composite tolerance. Every composite position tolerance has at least two segments.

Composite position tolerances are always formatted the same. One symbol is shown, and two or more segments are included. The first (upper) segment always specifies the *pattern-locating tolerance*. The second and any additional segments specify *feature-relating tolerances*. The pattern-locating tolerance is always a larger value than the feature-relating tolerance.

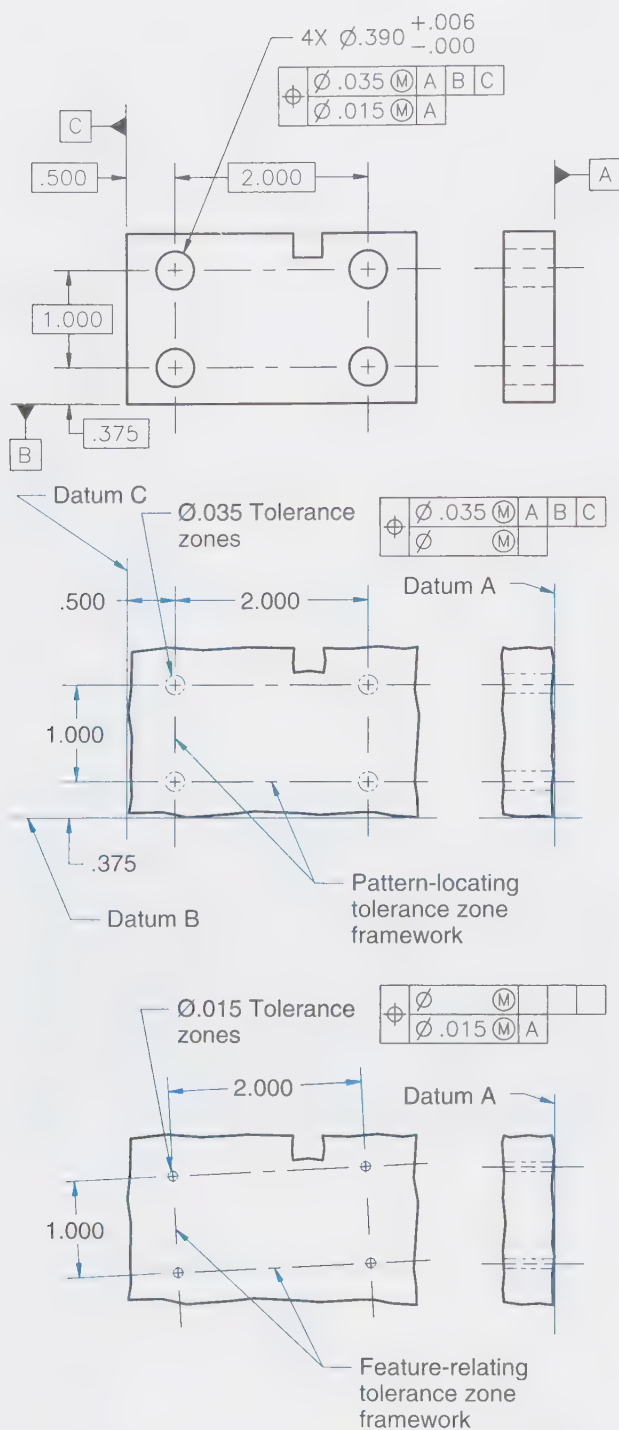
Figure 9-3 shows a simple plate with four holes in it. A composite position tolerance is applied to the four-hole pattern. The first segment specifies a .035” diameter pattern-locating tolerance. The second segment specifies a .015” diameter feature-relating tolerance.

The pattern-locating tolerance includes references to three datum features: datum feature A is primary, B is secondary, and C is tertiary. The feature-relating tolerance shown in the second segment only references primary datum feature A.

Note

Whenever datum references are shown in the second segment of a composite tolerance, they must be repeated from the first segment, in the same order of precedence, and with the same material boundary modifiers. There are no exceptions.

The pattern-locating tolerance of the first segment has the same effect as a single-segment position tolerance specification. Tolerance zones are located relative to the datum reference frame, at the true positions indicated by the basic dimensions. In the given figure, the tolerance zones have a diameter of .035”, assuming the holes are produced at MMC. The four tolerance zones are perfectly located (translation degrees of freedom are constrained) relative to the datum reference frame, and also have perfect orientation (rotation degrees of freedom are constrained) to the datum reference frame. The basic dimensions defining the true positions and the four pattern-locating tolerance zones are fixed in translation and rotation relative to the datum reference frame. This framework, which defines the true positions and is restrained in translation and rotation relative to the referenced datums, is known as the *pattern-locating tolerance zone framework (PLTZF)*, pronounced *plotz*.



Goodheart-Willcox Publisher

Figure 9-3. Each segment in a composite tolerance creates a tolerance zone framework that is sized by the basic dimensions that define the true positions for the holes.

The feature-relating tolerance of the second segment has a different effect than the pattern-locating tolerance. The second segment always defines the allowable feature-to-feature location variations and the allowable orientation variations between the features. The rotational degrees of

freedom relative to any datum feature referenced in the second segment are also constrained. However, the second segment does not constrain the translational degrees of freedom of the pattern of features relative to the datums. So, the second segment establishes a framework that includes true positions and feature-relating tolerance zones, and that framework is free to translate (move), provided that the framework does not rotate relative to the referenced datums.

On the given part, the second segment establishes tolerance zones located on true position axes on a 1.000" by 2.000" rectangle (a tolerance zone framework). The true position axes and tolerance zones are perpendicular to datum A because datum feature A is referenced. This 1.000" by 2.000" framework is basic and has no variation (no error). The true position axes of the holes are located at basic dimensions relative to one another, and this entire framework is free to translate (move) but not rotate relative to the referenced datum(s). The four feature-relating tolerance zones in their proper relative positions to each other and constrained from rotating relative to the specified datums is known as the *feature-relating tolerance zone framework (FRTZF)*, pronounced *fritz*.

The rotational degrees of freedom for the feature-relating tolerance zone framework are constrained relative to any datums referenced in the second segment. As previously stated, the second segment does not constrain translation of the framework relative to the datums. Location relative to the datums is only included in the pattern-locating tolerance of the first segment. This will be further explained using Figures 9-4 through 9-8.

A two-segment composite position tolerance provides two levels of control on tolerated features. The two levels of control permit the freedom to specify a relatively small tolerance to match hole patterns on mating parts and still permit a relatively large pattern-locating tolerance that affects edge distances. See **Figure 9-4**. The two requirements in the composite tolerance may be verified separately, but the requirements are not completely separate. Both requirements must be met by the same set of holes. So, although the feature-relating tolerance zone framework is free to translate relative to the referenced datums, there is a limit to the allowable amount of translation for the pattern of holes. That limit is defined by the pattern-locating tolerance. Where the holes meet both the pattern-locating tolerance and the feature-relating tolerance, the feature-relating tolerance zones will, at a minimum, overlap the pattern-locating tolerance zones.

In the given figure, the pattern-locating tolerance zone framework has the translation and rotation degrees of freedom constrained relative to a datum reference frame. Relatively large tolerance zones are located at the true positions defined by this framework.

Locations of the true positions and orientations of the true position axes relative to one another within the feature-relating tolerance zone framework are perfect. However, there is no requirement to fix the translational degrees of freedom of this framework relative to any of the specified datums. Only the rotational degrees of freedom of the framework are constrained relative to the datums specified in the second segment of the feature control frame. In the given figure, only the primary datum is referenced in the feature-relating tolerance. The relatively small feature-relating tolerance zones are centered on the true positions defined by this framework.

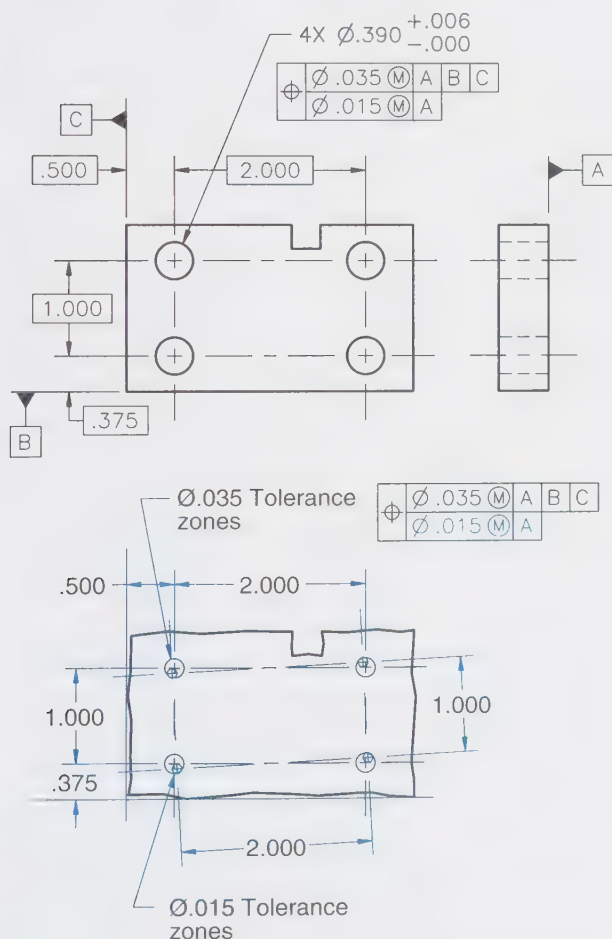


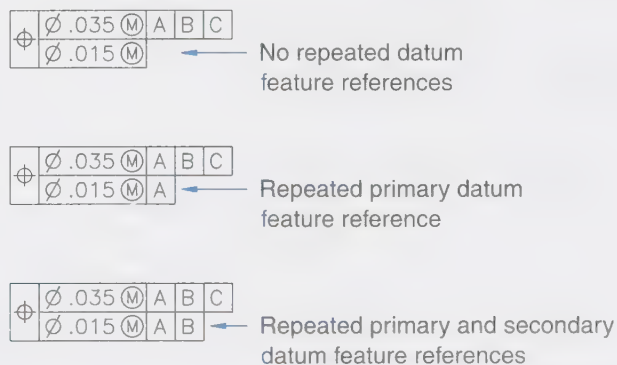
Figure 9-4. The feature-relating tolerances may move within and even extend outside the pattern-locating tolerance provided the centers of the holes are within all tolerance zones.

The feature-relating tolerance zone framework is free to translate relative to the datum reference frame. The cylindrical feature-relating tolerance zones move with the framework. The movement of the feature-relating tolerance zone framework is not directly controlled by the pattern-locating tolerance. It is permissible for the feature-relating tolerance zones to be within or overlap the pattern-locating tolerance zones. However, there is a practical limit to the allowable movement of the feature-relating tolerance zone framework, because the features must meet the requirements of both segments in the composite position tolerance specification.

Datum feature references are repeated in the second segment of the composite feature control frame to the extent necessary to achieve the desired level of control. See **Figure 9-5**. Any shown datum feature reference in the second segment must be made in the same order of precedence and with the same boundary modifiers as in the first segment. There are no exceptions. The feature-relating tolerance zone framework must have the rotational degrees of freedom constrained in relationship with any datum features referenced in the second segment.

No Repeated Datum References

Absence of any datum reference in the second segment of a composite tolerance creates a feature-relating tolerance zone framework that is not constrained in translation or rotation. It is free to float relative to the datum reference frame. See **Figure 9-6**. A composite position tolerance is shown. The first segment specifies a .025" diameter pattern-locating tolerance, and it references datum

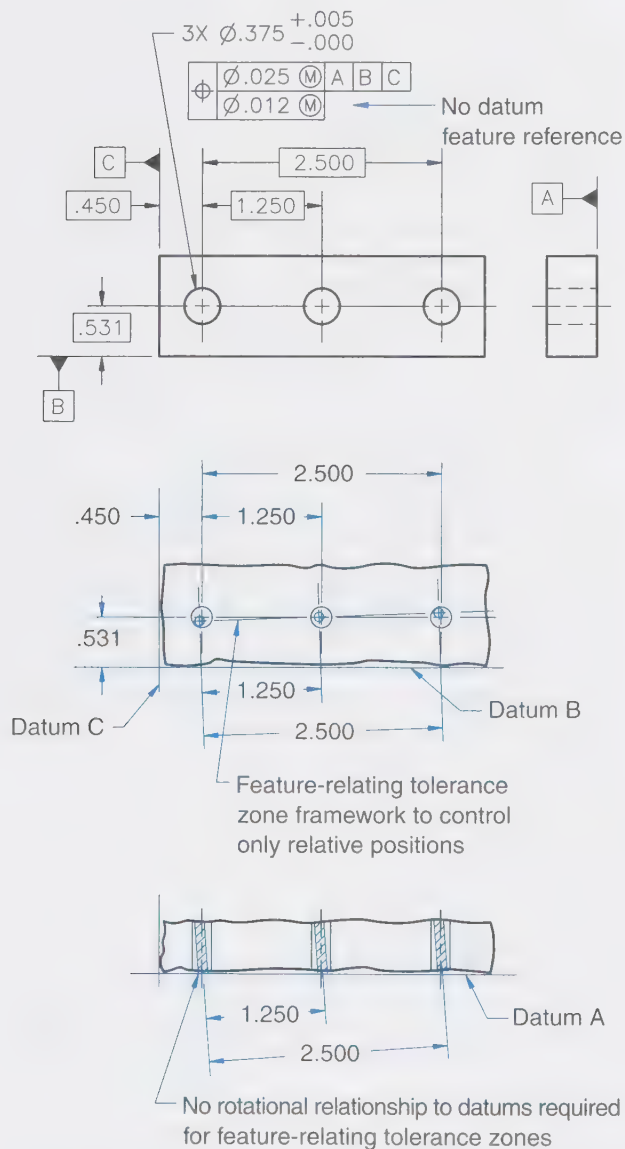


Goodheart-Willcox Publisher

Figure 9-5. The number of datum feature references repeated in the second segment of a composite tolerance is determined by the desired rotational degrees of freedom to be constrained on the feature-relating tolerances.

features A, B, and C. The second segment specifies a .012" diameter feature-relating tolerance zone that does not reference any datum features.

The pattern-locating tolerance zone framework creates .025" diameter tolerance zones located on the true positions dimensioned from the datum reference frame. The feature-relating tolerance zone framework creates .012" diameter tolerance zones that are relative to one another at the basic dimensions defined on the drawing. The feature-relating tolerance zone framework is free to translate and rotate because no datums are referenced in the second segment.



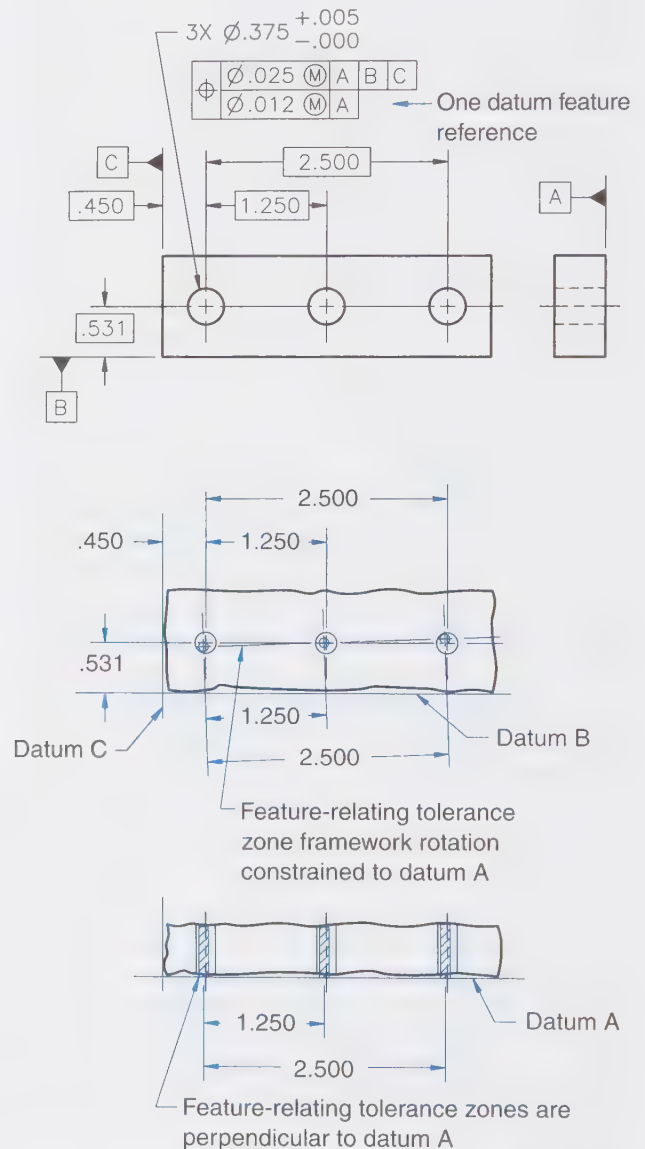
Goodheart-Willcox Publisher

Figure 9-6. Omitting all datum references from the second segment constrains no rotational degrees of freedom for the feature-relating tolerance. The second segment of the composite tolerance only controls feature-to-feature locations.

The feature-relating tolerance zones are not required to be completely contained within the pattern-locating tolerance zones. However, holes produced in the part must be able to meet the tolerance requirements of both the first and second segments of the feature control frame.

Repeated Primary Datum Reference

Repeating the primary datum reference in the second segment of a composite tolerance creates a feature-relating tolerance zone framework that has the rotational degrees of freedom constrained relative to the primary datum. See [Figure 9-7](#). The feature-relating tolerance zone framework is free



Goodheart-Willcox Publisher

Figure 9-7. The primary datum feature reference constrains the rotational degrees of freedom for the feature-relating tolerance zone framework, relative to the primary datum.

to translate relative to the datum reference frame so long as the rotational degrees of freedom relative to the primary datum are maintained.

A composite position tolerance is shown in the given figure. The first segment requires a .025" diameter tolerance on the pattern-locating tolerance zone framework, which is referenced to datum features A, B, and C. The feature-relating tolerance is .012" diameter and is referenced to datum feature A.

The pattern-locating tolerance zone framework creates .025" diameter tolerance zones located on the true positions dimensioned from the datum reference frame. The feature-relating tolerance zone framework creates .012" diameter tolerance zones that are located relative to one another at the basic dimensions defined on the drawing. The rotational degrees of freedom of this framework are constrained relative to datum A, but translation is not constrained.

Holes produced in the part must be able to meet both tolerance requirements.

Repeated Primary and Secondary Datum Reference

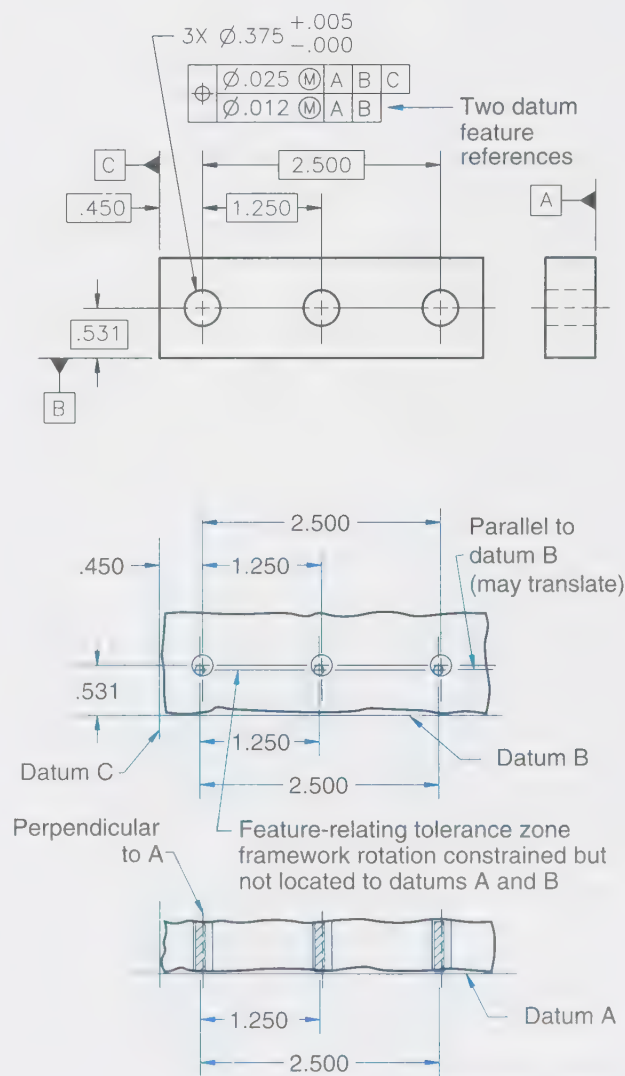
The 1994 and current standards clearly illustrate composite position tolerances that include two datums in the second segment. Drawings containing composite position tolerances that show primary and secondary datum feature references no longer require supplemental notes to explain the tolerance requirements. Drawings created prior to the 1994 standard may include clarifying notes where secondary datum feature references are shown in the second segment of a composite position tolerance.

If a drawing is being completed in compliance with 1982 or earlier standards, it is advisable to include a note to define the interpretation of any shown composite tolerances. ANSI Y14.5M-1982 and earlier standards did not clearly illustrate the interpretation of multiple datums in the second segment, so some controversy regarding the correct interpretation may exist if no note is shown. The following is a suggested note:

THE SECOND SEGMENT OF A COMPOSITE POSITION TOLERANCE CONTROLS RELATIONSHIPS OF FEATURES WITHIN A PATTERN. DATUM REFERENCES IN THE SECOND SEGMENT OF A COMPOSITE POSITION TOLERANCE CONSTRAIN THE ROTATIONAL DEGREES OF FREEDOM FOR THE TOLERANCE ZONE FRAMEWORK RELATIVE TO THE REFERENCED DATUMS BUT THE TRANSLATIONAL DEGREES OF FREEDOM ARE NOT CONSTRAINED.

Repeating the primary and secondary datum reference in the second segment of a composite tolerance creates a feature-relating tolerance zone framework that is fixed in rotation relative to the two referenced datums. See Figure 9-8. The datum references in the second segment do not constrain the translational degrees of freedom relative to the referenced datums. The feature-relating tolerance zone framework is free to translate relative to the datum reference frame so long as the rotational degrees of freedom relative to the primary and secondary datums are maintained.

A composite position tolerance is shown in the given figure. The first segment requires a .025" diameter tolerance on the pattern-locating tolerance zone framework, which is referenced to



Goodheart-Willcox Publisher

Figure 9-8. Including both the primary and secondary datum feature references constrains the rotational degrees of freedom for the feature-relating tolerance zone framework relative to the two referenced datums.

datum features A, B, and C. The feature-relating tolerance is .012" diameter and is referenced to datum features A and B.

The pattern-locating tolerance zone framework is fixed in translation and rotation relative to the datum reference frame and has .025" diameter tolerance zones located on the true positions for the holes. The feature-relating tolerance zone framework has .012" diameter tolerance zones that are located relative to one another at the basic dimensions defined on the drawing. The rotational degrees of freedom for this framework are constrained (or fixed) relative to datums A and B (perpendicular to A, parallel to B). The figure shows one possible location of the feature-relating tolerance zone framework. The framework is permitted to translate up, down, left or right, relative to datum B, so long as it remains parallel to datum B. Of course, the translation of the framework is limited because the hole positions must also meet the requirements of the first segment.

Verification of Composite Tolerances

One set of measurements may be taken for verification of composite position tolerances when a computer program or manual calculation process is used to assess the variations relative to the pattern-locating tolerance and also the feature-relating tolerance. At least four United States patents are related to the processes to do this. Bruce A. Wilson, the author of this text, is the sole inventor for two of those patents and coinventor of another. The patents based on Mr. Wilson's inventions are owned by The Boeing Company.

The above mentioned processes may only be used under license from the Boeing Company, and because the calculation complexities are beyond the scope of this book, they are not detailed here. However, anyone desiring to know more about those processes may look up the patents through the US Patent and Trademark Office website because patents are public records. However, using the processes in the patents would require license from the owner. When creating new processes or designs, it is always a good idea to perform a patent search to avoid the consequences of patent violation.

The following explanation of verification methods for composite position tolerances is common knowledge within the industry and provides adequate insight for the reader to be able to understand composite position tolerances. These processes will also permit verification of tolerances on produced parts.

A paper gaging process may be used for composite position tolerances. This process is similar

to the process already explained for single-segment position tolerances. Two sets of measurements are typically taken when using manual inspection methods. The first set of measurements determines the locations of the holes relative to the datum reference frame. Another set determines the locations of the holes relative to each other.

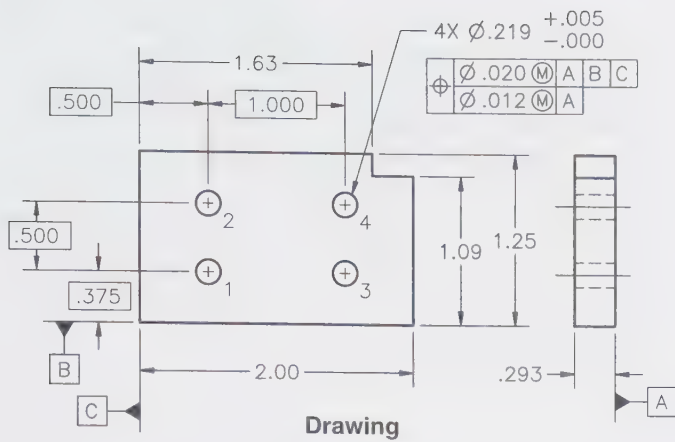
Figure 9-9 shows a four-hole pattern with a composite tolerance applied to it. Data from a produced part is entered into the two tables. Inspection data may be received in a format that is not easily utilized, and in such cases putting that data in a table can make it easier to complete the evaluation of the information. The tables shown in the given figure are formatted for the convenience of explaining concepts and to fit the available space. They may not be optimum for handling large amounts of data.

Data in the first table shows measured hole locations relative to the datum reference frame. This data is used to verify compliance with the pattern-locating tolerance. The measured coordinate location variations are used to determine the diameter of positional variation that exists. This may be done either by calculation or by plotting the coordinate variations on a grid with concentric circles representing the position tolerance zones. The figure shows the X and Y variations plotted on a grid and concentric circles representing the specified pattern-locating tolerance and the allowable bonus tolerance.

Data in the second table shows the measured hole locations relative to one another. This data is used to verify compliance with the feature-relating tolerance. Measurement of the relative hole locations requires that two holes be used to establish a coordinate system. The measured relative positions could vary in magnitude depending on which two holes are selected to establish the coordinate system. Any two holes may be used for the initial measurements. In the figure, hole #1 is used as the origin, and hole #3 is used to establish the orientation of the X axis.

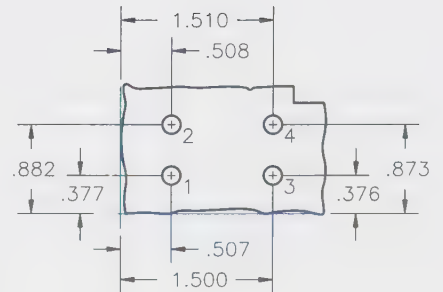
It is not important which holes are selected to establish the initial coordinate system. If measurements indicate a hole is too far out-of-position, another set of holes may be used to establish a new coordinate system. Any two holes may be used that will result in the optimum set of measurements.

The coordinate variations for each hole location are plotted on a grid. The hole that acted as the origin for the measurements is plotted at the 0,0 coordinate of the grid. In this figure, hole #1 is placed at the 0,0 coordinate. The coordinate variations of the other holes are plotted relative to the location of hole #1.



Hole location coordinate variation from datums

Hole #	1		2		3		4	
Diameter	.222		.223		.221		.223	
	X	Y	X	Y	X	Y	X	Y
Measured location	.507	.377	.508	.882	1.500	.376	1.510	.873
Drawing dimension	.500	.375	.500	.875	1.500	.375	1.500	.875
Variation	.007	.002	.008	.007	.000	.001	.010	-.002



Hole-to-hole location variance

Hole #	1		2		3		4	
Diameter	.222		.223		.221		.223	
	X	Y	X	Y	X	Y	X	Y
Measured location	0	0	.0017	.5048	.9930	0	1.0024	.4956
Drawing dimension	0	0	0	.500	1.000	0	1.000	.500
Variation	0	0	.0017	.0048	-.0070	0	.0024	-.0044

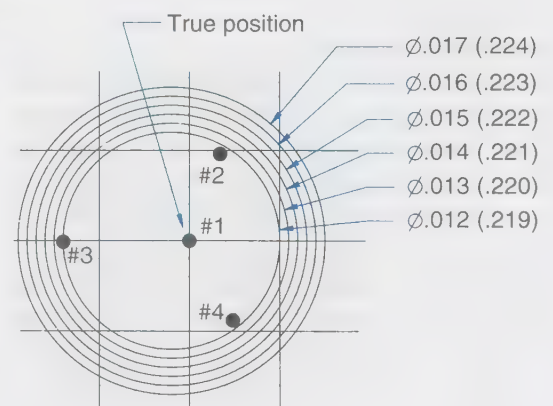
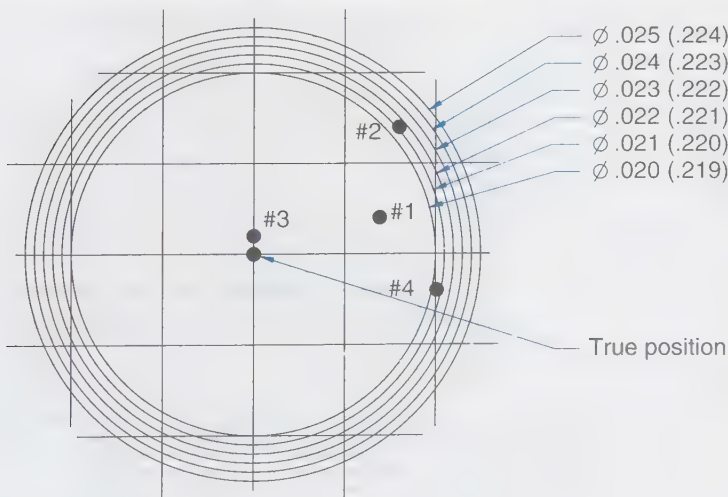
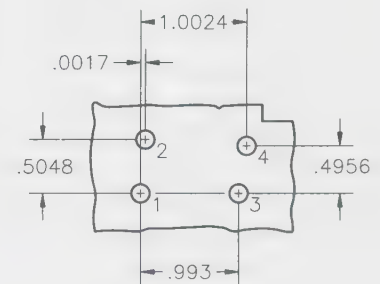


Figure 9-9. Coordinate hole location variations can be plotted on graph paper and an overlay of concentric circles can be used to determine positional tolerance variation for the holes.

A set of concentric circles representing tolerance zone diameters is placed over the plotted hole locations. There is no requirement for the circles to be centered on the origin because the feature-relating tolerance zone framework is free to translate relative to the datum reference frame.

Each concentric circle represents a specific diameter tolerance zone, and that diameter is related to a produced hole diameter. The concentric circles may be moved into any position, provided the center point for each hole is contained within the circle that corresponds to the produced hole diameter. Movement of the concentric circles is representative of the feature-relating tolerance zone framework translation relative to the datum reference frame.

The paper gaging process will not accept a part that violates the allowable positional tolerance. However, the process may result in rejection of a part that is marginally good. This can happen because the origin for measurements set at the center of one of the holes and axis direction is established through the center of another hole. This constrains the degrees of freedom during measurement in a way that is not identical to a mating part that is free to move to a best fit position. Another limitation is the iterative nature of selecting the holes to establish the coordinate system. If the position variation for one or more of the features is slightly larger than the specified feature-relating tolerance, it is not known that the part should be rejected until all possible hole combinations have been used to create all possible coordinate systems that may result in acceptable measurements.

It is reasonable to ask why this process should be known if it sometimes rejects acceptable parts. First, it is a relatively easy means of explaining composite position tolerance requirements. Also, it is a relatively easy process to use and is a good means for verifying that parts are acceptable. It may also be used to reject parts if the feature-relating variations are significantly larger than the specified tolerance. Finally, a simple process may save enough cost to justify the risk of rejecting a small percentage of marginally acceptable parts.

Functional gages may be designed that will verify the feature-relating tolerance. This will be further explained in this chapter. The functional gage confirms the position tolerance is met, but there is no indication of the amount of the available tolerance that was consumed.

When hole patterns are relatively small or the datum features are small, there is an increased likelihood for the feature-relating tolerance zone

framework to have some rotational variation relative to the datums. The rotational variation is the reason that it is necessary to establish a coordinate system from features in the pattern and to take measurements using that coordinate system.

Experience has shown that as the datum features increase in size and the hole pattern gets larger, the rotational variation of the feature-relating tolerance zone framework relative to the datums may become insignificant. When the rotational variation of the feature-relating tolerance zone framework is negligible, it is possible to use one set of measurement data for paper gaging both the pattern-locating tolerance and the feature-relating tolerance. This method is safe for accepting parts but does have some risk of rejecting acceptable parts. The risk increases as part and hole pattern sizes get smaller.

Two Single-Segment Feature Control Frames

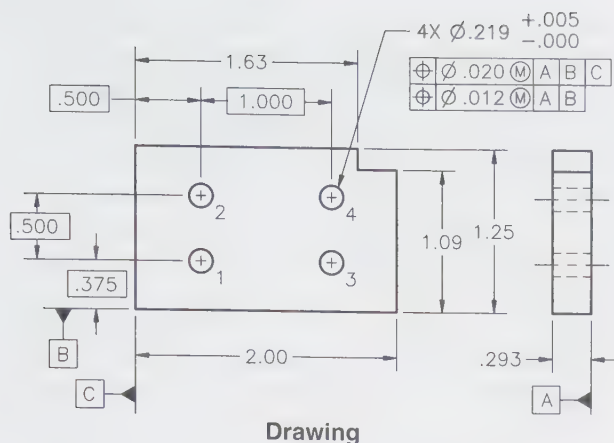
Two single-segment position tolerance feature control frames show two position tolerance symbols and have a different meaning than a composite position tolerance. See **Figure 9-10**. The two single-segment specifications establish two tolerance zone frameworks, and each is fixed (constrained) in translation and rotation relative to the datum reference frame.

In the given example, segment one requires a position tolerance of .020" diameter relative to three datums: A, B, and C. The tolerance zone framework and the associated .020" diameter tolerance zones are located by basic dimensions from the referenced datums. The tolerance zones are also oriented relative to the datums.

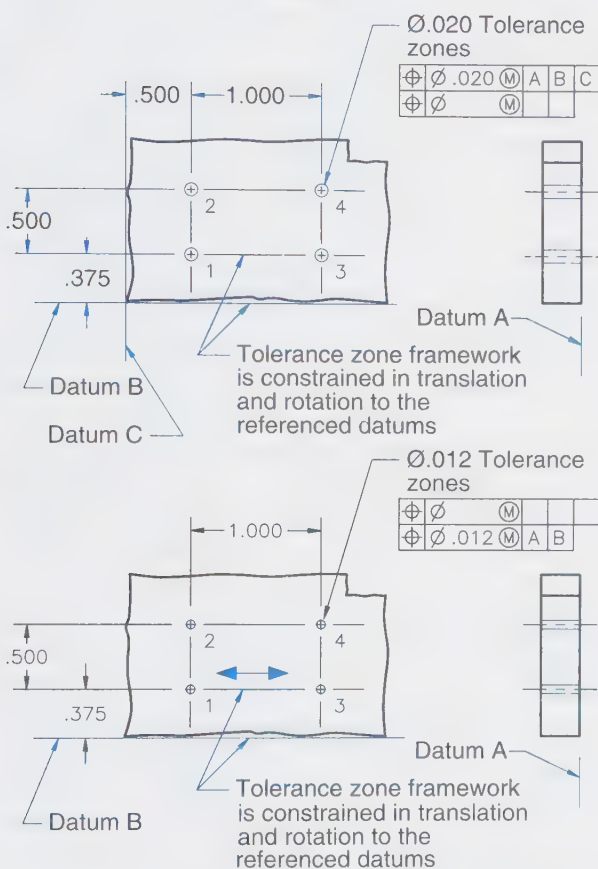
The second segment requires a position tolerance zone of .012" diameter relative to two datums: A and B. The tolerance zone framework and the associated .012" diameter tolerance zones are located by basic dimensions from the referenced datums A and B. The tolerance zones are also oriented relative to the referenced datums. There is no location or orientation requirement relative to the omitted datum C.

Functional Gaging of Position Tolerances

Functional gages may be used to efficiently inspect positional tolerances that are specified at MMC. Functional gages are tools used to verify that produced parts meet tolerance requirements. Gages for some applications are very simple.



Drawing



Goodheart-Willcox Publisher

Figure 9-10. Using two position tolerance symbols creates two single-segment position tolerance specifications. Each of the segments creates an independent tolerance zone framework.

Simple gages are inexpensive to design and produce. The cost of gages can be quickly recovered when they are used to reduce the amount of time spent performing labor-intensive manual measurements.

Complex parts and tolerances impact the complexity of functional gages. Complex gages require more design and fabrication time than simple gages. The cost of a complex gage must

be balanced against the cost of manual measurements, the use of computer-driven measurement machines, or even the utilization of process controls that require little or no inspection. If a part is complex and difficult to check manually, the functional gage may pay for itself through time saved during inspection, even if the gage is expensive to design and produce.

The information contained in the following paragraphs is provided to explain what functional gages are, and to demonstrate through example gages what the workpiece tolerance specifications require. Gage dimensions in the given examples are determined from the workpiece dimensions and tolerances. No gage design tolerances are calculated or shown. The inclusion of gage tolerance calculations in the examples would complicate the explanations of what the functional gages are designed to verify. Also, the subject of gage calculations is extensive and beyond the scope of this book.

It should be kept in mind throughout this section that the datum simulation features in the gages are physical representations of perfect true geometric counterparts of the datum features. The simulators in the figures are illustrated as perfect and therefore do not introduce any variation into the explanations of tolerances that are applied relative to datum features.

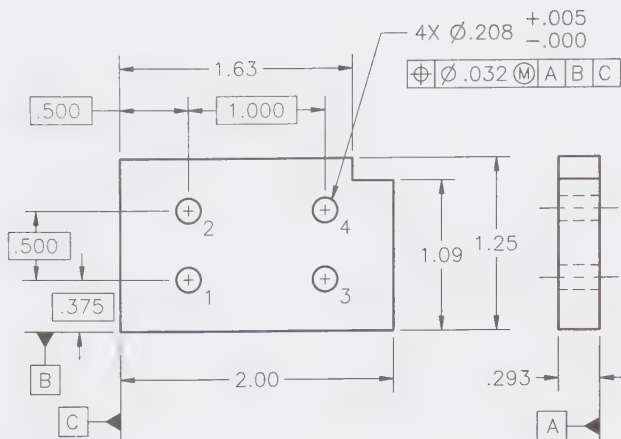
It is also important to realize that functional gaging (the use of physical gages and datum simulators) is not required to verify positional tolerances. The same concepts explained through the gaging explanations may be accomplished through metrology and mathematical processes.

Real-world application of physical simulators and simulation processes does introduce the variation of the simulation process (somewhat like measurement error) into the inspection process. That variation must be considered in the verification process. The variation that is present in the physical simulation process is often subtracted from the allowable tolerances on the features.

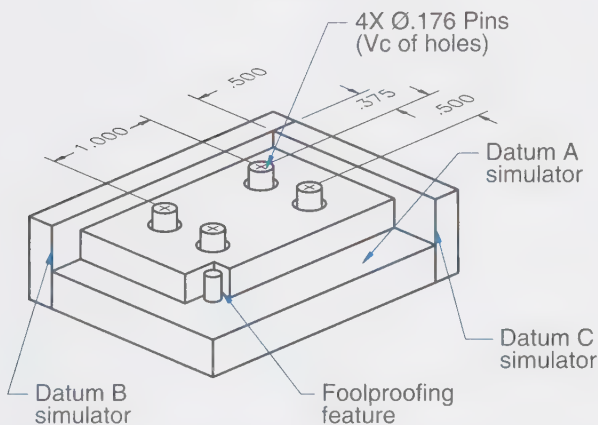
Gaging for Tolerances Specified at MMC

Position tolerances can be functionally gaged when the tolerances include an MMC modifier. See [Figure 9-11](#). The given figure includes a drawing of a plate that contains four holes. A positional tolerance including an MMC modifier is applied to the holes.

A functional gage for checking hole locations produced to the given drawing must include features to properly locate the datum reference frame and to check the locations of the holes relative to that datum reference frame. The shown position



Drawing



Goodheart-Willcox Publisher

Figure 9-11. Functional gages may be used to check the location requirements defined by position tolerances.

tolerance on the four holes includes a reference to datum feature A primary, B secondary, and C tertiary. To properly locate the part in the gage, there must be datum simulators for all three datums. The gage shown in the figure has three plates that correctly simulate the datums.

Four pins are installed in the gage to check the locations of the holes. The pins are located at the true positions specified on the drawing. Assuming the gage can be made perfect (theoretically exact), the pins are sized at the virtual condition of the holes. The virtual condition is equal to the hole at MMC minus the position tolerance.

A part produced to the given drawing can be inspected to verify the position tolerance for four hole positions by placing the part over the four pins. In addition to fitting over the four pins, several conditions must be met to verify that the holes are in acceptable positions. The part must rest against the three datum simulators, properly establishing the datum reference frame, with all four pins extending through the holes. To properly establish the datum reference frame, at least three

points must be in contact with the primary datum simulator, two with the secondary datum simulator, and one with the tertiary datum simulator.

The shown gage is designed to check the hole positions based on the virtual condition of the holes (.208 MMC hole - .032 position tolerance = .176 virtual condition pin). The gage verifies that the surfaces of the holes have not violated the virtual condition boundaries at the true positions, so the gage is using the surface method of verifying the hole locations are acceptable. Sizing the pins at the virtual condition results in an adequate check of position for any hole that is produced within size tolerance. The fixed diameter pins automatically permit the allowable bonus tolerance when a hole is produced at any size larger than MMC. The shown gage does not verify hole size. Hole diameter must be checked separately. The gage does not determine the amount of variation in the hole locations; it only verifies that the position tolerance has not been exceeded.

Gaging When a Primary Datum Is Referenced at MMB

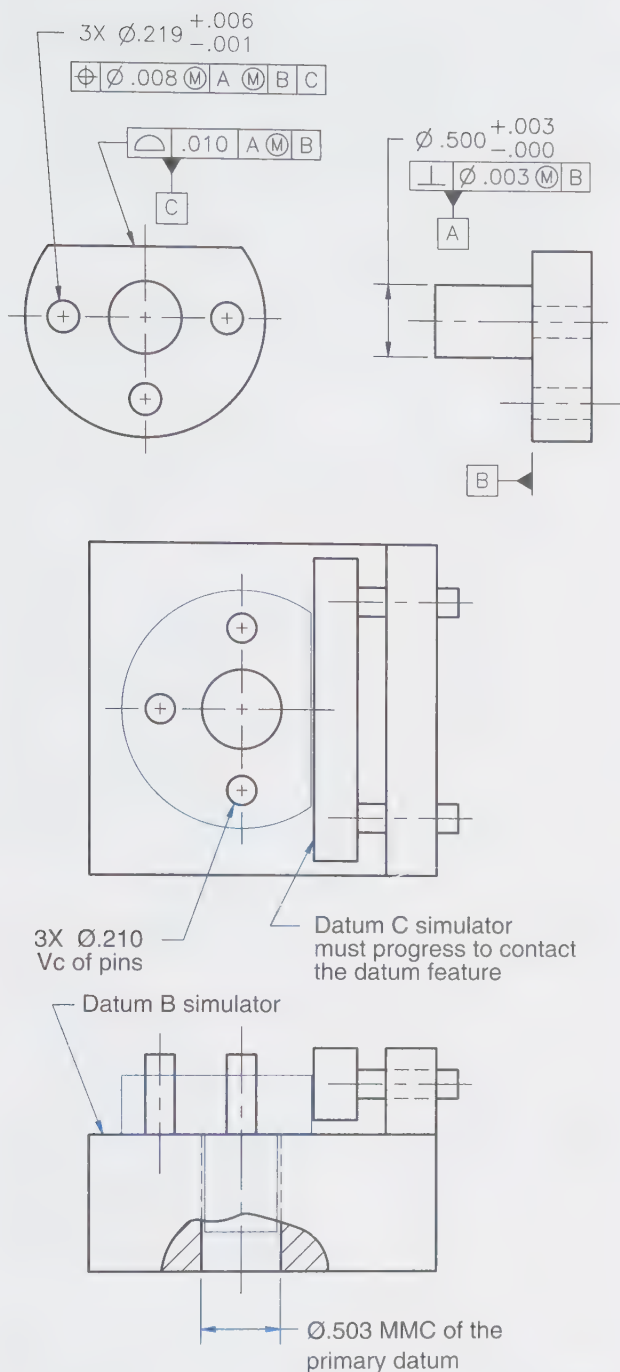
References to a datum feature of size in a position tolerance must include a material boundary modifier. Material boundary modifiers applied to datum references have a significant impact on how the datum reference frame is established. Generally, the application of an MMB modifier makes functional gaging easier to accomplish.

Figure 9-12 includes a drawing of a simple part with a .500" diameter shaft identified as datum feature A. Three holes have a position tolerance specified that references primary datum feature A, and the MMB modifier is applied.

Because primary datum feature A is a cylinder, a datum axis is established and two planes of the datum reference frame intersect on that axis. The third plane is established by secondary datum feature B (a surface). Datum feature C (a surface) is referenced only to establish a rotational position for the hole pattern.

There are special rules about how a datum feature of size must be established. One of the rules applies to a primary reference to a datum feature of size. When a primary datum reference to a datum feature of size includes the MMB modifier, and no form tolerance is applied to the feature of size, the datum feature simulator is fixed in size at the maximum material condition size of the datum feature.

A functional gage for the given part includes a hole to pick up the datum feature. The simulated datum axis is at the center of the hole. The hole



Goodheart-Willcox Publisher

Figure 9-12. A primary datum referenced at MMB is simulated at the MMC size of the datum feature when no form tolerance is applied to the primary datum feature.

is sized at the .503" maximum material condition size of the shaft. The shaft would fit snugly (or be a press fit) into the hole when the shaft is at the MMC size, and any shaft produced at a diameter smaller than MMC size will be free to float within the gage. The .503" shaft in the .503" diameter hole would result in the datum axis and the feature axis coinciding. If the shaft is smaller than .503"

diameter, the clearance between the shaft and the simulator permits relative movement between the feature axis and simulated datum axis.

With the primary datum feature in the .503" hole in the gage, secondary datum feature B is moved into contact with the top surface of the gage. Depending on how tightly the shaft fits into the hole and the amount of perpendicularity variation on the shaft, the datum feature B surface may make only one point contact. An adjustable plate is designed to progress through the profile tolerance zone for datum feature C and will press up against datum feature C to establish the correct rotational position of the part.

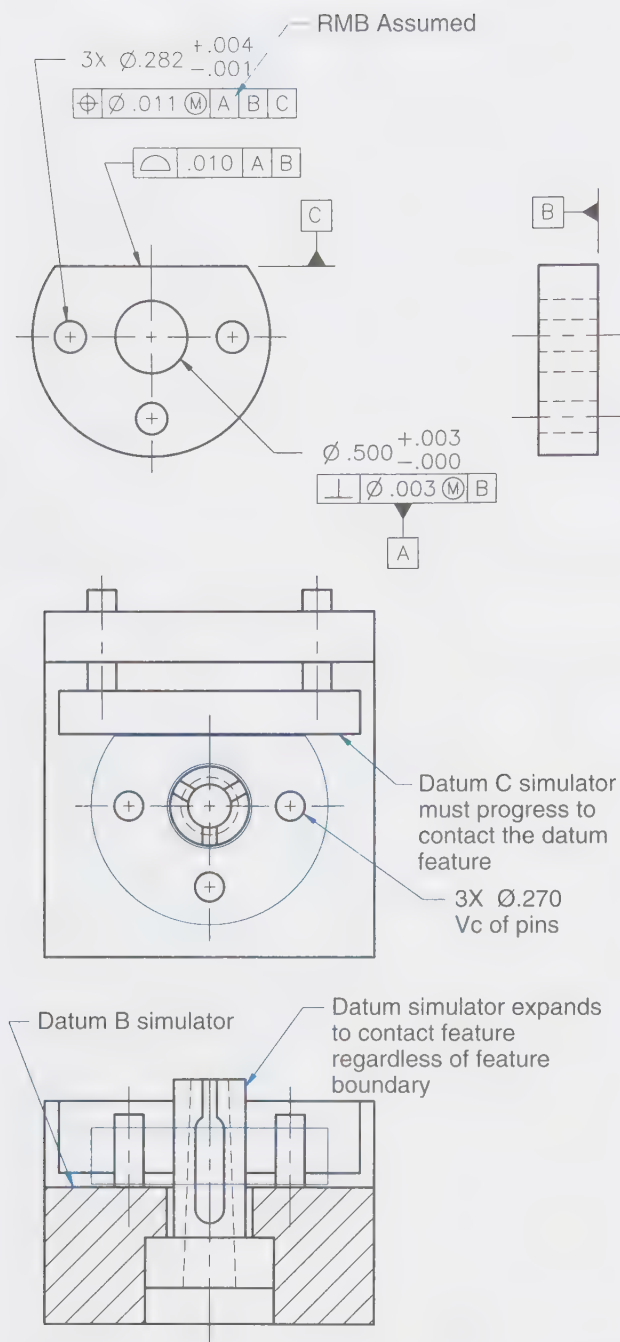
The three pins in the gage are sized at the virtual condition of the holes. The pins must extend through the holes while datum features A, B, and C are properly located in the gage.

Gaging When a Primary Datum Is Referenced at RMB

References to datum features of size at RMB can complicate gage design, but the requirement can often be accurately simulated in a gage. See **Figure 9-13**. The given figure includes a drawing of a circular plate with four holes in it. The center hole is identified as datum feature A. The three hole pattern has a position tolerance that references datum feature A primary. No modifier is shown, so RMB is applicable.

Because datum feature A is primary and referenced at RMB, a datum simulator that actually makes contact with the part surface must be used. The size of a cylinder that makes actual contact is called the *actual mating envelope*. Because this envelope is not related to a higher precedence datum, it is more specifically known as an *unrelated actual mating envelope*. Contact between the simulator and the datum feature is required regardless of the produced diameter of the hole. Adequate contact must be made to align the axis of the datum simulator and the axis of the produced hole.

The gage for this part has an expanding pin that makes contact with the primary datum. It is expanded enough to align the axis of the gage pin with the produced hole. With the pin properly expanded, the part is moved down the pin until it makes contact with the datum B simulator. Datum feature B may make only one point contact with its simulator. With the relationship between the part and two datum simulators already established, the tertiary datum simulator progresses through the profile tolerance zone into contact with datum feature C to establish a fixed rotational position of the part.



Goodheart-Willcox Publisher

Figure 9-13. A primary datum feature referenced at RMB is simulated by a gage feature that makes contact with the datum feature.

Three pins sized at the virtual condition of the holes are passed through the holes while the part is located on the datum simulators.

Effect of Secondary and Tertiary Datum References at MMB

Each datum feature of size that is referenced with the MMB modifier requires a fixed size datum feature simulator (physical or mathematical) to

establish the appropriate datum. Depending on whether a feature has an orientation, profile, or position tolerance applied to it, there may be more than one MMB for the feature. The size of the datum feature simulator is determined by the applicable MMB of the datum features as determined by the tolerances applied to the feature relative to higher precedence datums.

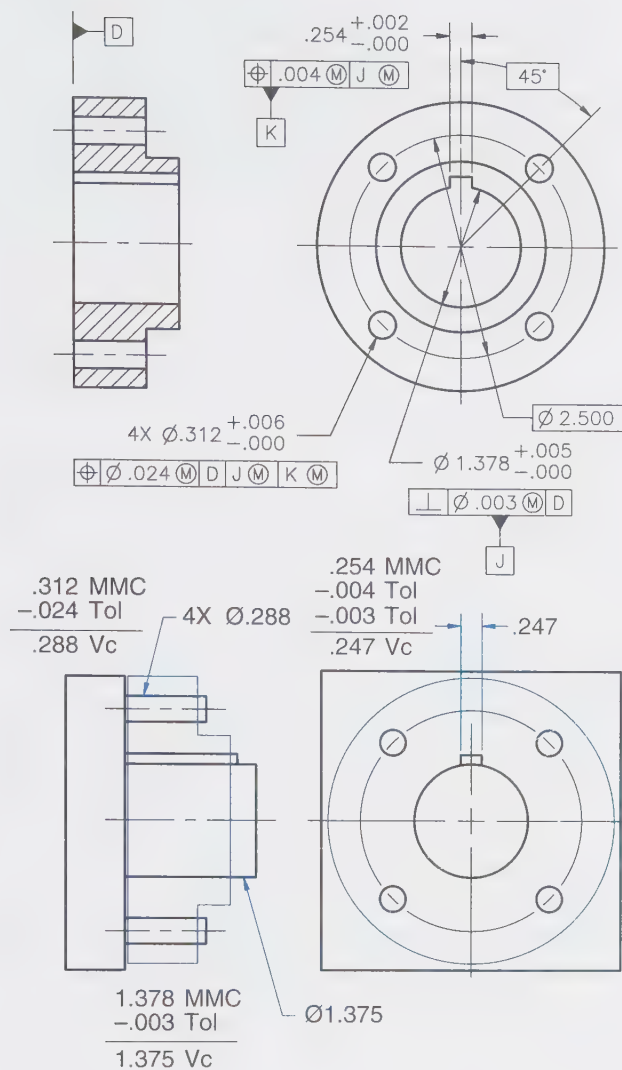
The applicable MMB for a feature of size is the same as the applicable virtual condition. As stated above for the applicable MMB, applicable maximum material boundary (applicable virtual condition) is determined by the tolerances applied to the feature relative to higher precedence datums. Sizing the datum simulator equal to the virtual condition of the feature is essential to achieve the proper level of constraint of the datum feature relative to the datum simulator.

It is incorrect to simulate a secondary or tertiary datum at the maximum material condition size except when a geometric tolerance of zero at MMC is applied to it. The virtual size for a datum feature of size is the worst case combined effect of the size tolerance and the geometric tolerances that are referenced to higher precedence datums. The virtual condition can be very difficult to determine if the geometric tolerances on a datum feature are not referenced to higher precedence datums. Although Figure 9-12 showed an example of a position tolerance where a higher precedence datum (A) is qualified to a lower precedence datum (B), this is unusual and should only be done when it will not impact the ability to properly establish the datum reference frame.

Figure 9-14 illustrates how secondary and tertiary datum references made at MMB impact the datum simulators. Several aspects of the given drawing must be observed before the gage requirements can be explained.

The given part has a large diameter hole bored through the center, and the hole has a keyseat. A perpendicularity tolerance of .003" diameter at MMC is permitted on this hole, relative to datum D. The hole is identified as datum feature J. The keyseat has a position tolerance of .004" at MMC relative to datum J at MMB. The four small holes have a position tolerance of .024" diameter at MMC relative to datum feature D primary, datum feature J secondary at MMB, and datum feature K tertiary at MMB.

Application of MMB modifiers on the datum references makes it possible to use a single gage to check some of the datum feature requirements, and also to check the hole locations. This means that one gage can be designed to check the .024"



position tolerance on the four holes and at the same time verify some of the tolerances on the referenced datum features of size.

The given figure shows a gage for checking the hole locations and the datum features. Primary datum feature D is a flat surface. Its true geometric counterpart is a plane and is simulated with a simple flat plate. Secondary datum feature J is a hole. Its true geometric counterpart is a cylinder of fixed diameter and at a fixed orientation to the higher precedence datum. A pin perpendicular to datum D and sized to the virtual condition of the datum hole is used to simulate datum J. In addition to establishing the location of datum J, this pin will verify the perpendicularity requirement on the hole. Tertiary datum K is a keyseat. The true geometric counterpart is a pair of planes at a fixed

width and a fixed location and orientation relative to the higher precedence datums. A key sized to the virtual condition of the keyseat is used as the tertiary datum simulator.

Primary datum feature D is a simple flat surface. It locates a primary datum plane. Secondary datum feature J is a hole. It establishes the location of the secondary datum axis. This datum axis must be perpendicular to the primary datum plane, so the simulator only determines the location of the axis. Two of the planes in the datum reference frame intersect on the secondary datum axis. These two planes must be perpendicular to the primary datum plane because the datum axis is perpendicular to the primary plane.

Although the two planes located by datum feature J must be perpendicular to the primary datum plane, the hole itself can have a perpendicularity variation of .003" when the hole is at MMC.

The datum simulator for referenced secondary datum J at MMB must be sized to the virtual condition that results from the size and perpendicularity tolerance on that datum feature. This permits the hole to be produced with the specified amount of variation, and still have the datum reference frame be perfect. Because the datum simulator is smaller than the MMC size of the feature, the feature can have some variation and still fit into the gage. Proper installation of the workpiece in the gage requires that the primary datum maintain at least a three point contact with its simulator when the secondary datum simulator is inserted into the secondary datum feature.

For the given example, the pin in the gage must have a 1.375" diameter. This is the virtual condition for datum feature J. The 1.375" diameter virtual condition is determined by calculating the combined effect of the MMC size and the perpendicularity tolerance applied to the feature. In the given example, the virtual condition is equal to:

$$1.378'' - .003'' = 1.375'' \text{ diameter}$$

It is incorrect to size the pin in the given gage to the MMC size of the secondary datum. The result of such an incorrectly designed gage would be incorrect simulation of the datums. If the pin is equal to the MMC size of the hole, it could incorrectly act as the primary datum. A gage pin incorrectly sized at MMC size for the hole will act as a primary datum simulator if the hole in the given part is produced at MMC and has a .003" perpendicularity variation.

Tertiary datums referenced at MMB must also be properly simulated to establish the datum reference frame. The tertiary datum reference in the

given figure includes an MMB modifier. The gage must therefore simulate the tertiary datum at its applicable virtual condition.

Tertiary datum K is the keyseat on the part. The position tolerance on the four holes references datum features D primary, J at MMB secondary, and K at MMB tertiary. The key in the gage that simulates datum K is sized at the virtual condition of the keyseat relative to the primary and secondary datums for the position tolerance on the holes. The keyseat has a position tolerance that references datum feature J and datum feature J has a perpendicularity tolerance relative to datum feature D. The virtual condition for this feature is impacted by both the orientation tolerance on the 1.378" diameter hole and the position tolerance on the keyseat. Sizing the simulator to the virtual condition caused by both the applicable tolerances will permit the gage key (simulator) to enter the slot if all features are produced within the specified tolerance.

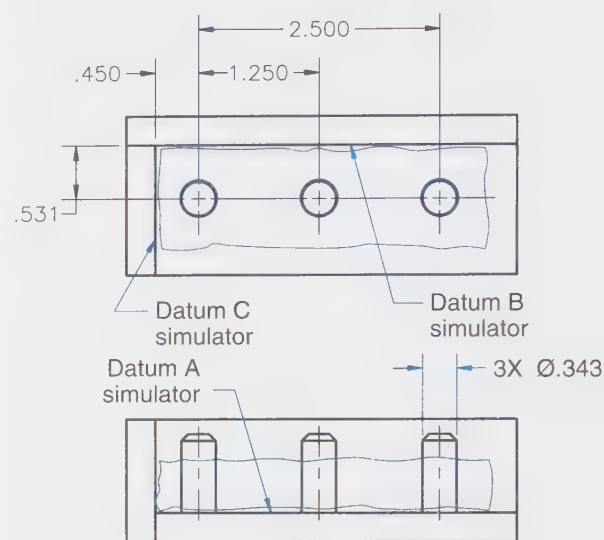
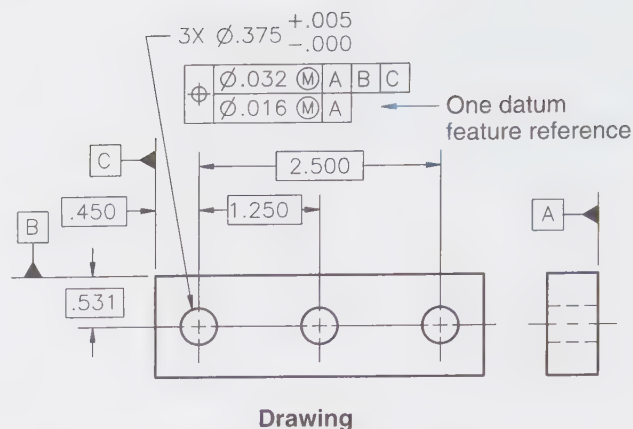
Incorrectly producing a gage with a key sized to the MMC size of the slot might incorrectly establish the datum reference frame. A key sized to the MMC size of the keyseat would not enter a keyseat that is produced at MMC unless the keyseat is perfectly located to the hole and the hole is perfectly oriented to the primary datum.

Position of the four .312" diameter holes can only be verified if the datum reference frame is properly established. The hole locations are checked by pins that are located at the true positions relative to the datum reference frame created by the datum simulators.

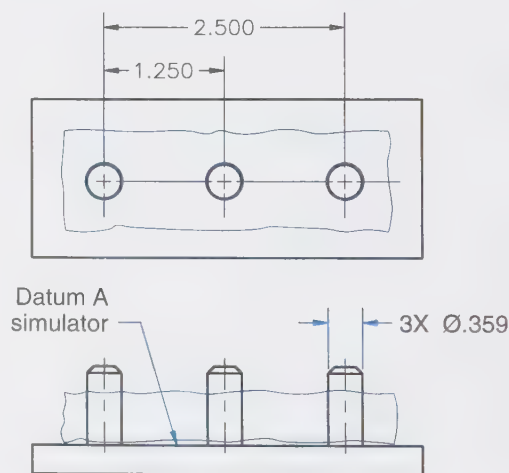
In the gage, pins for checking hole locations are sized at the virtual condition of the holes. Pins used to check hole locations are always sized to the virtual condition if the position tolerance includes the MMC modifier. The pins will verify that the surface of each hole has not violated the virtual condition boundary.

Gaging a Composite Position Tolerance

Composite position tolerances are gaged in a similar manner to the single-segment position tolerance specification. See [Figure 9-15](#). The given example shows a drawing with a composite position tolerance and the two gages used to check the produced hole locations. It is possible to design a single gage that will check both segments of a composite position tolerance, but two separate gages better illustrate how the hole positions may be checked.



Gage for pattern-locating tolerance



Gage for feature-relating tolerance

Goodheart-Willcox Publisher

Figure 9-15. Separate gages may be used to check the requirements for the two segments of a composite position tolerance.

The given composite tolerance specification requires a pattern-locating tolerance of .032" diameter at MMC relative to datum feature A primary, B secondary, and C tertiary. The gage used to check the pattern-locating segment of the composite tolerance is the same as it would be for a single-segment position tolerance specification. Plates are used to simulate the three datum planes. Pins are located at the true positions for the holes, and the pins are sized to the virtual condition created by the .032" position tolerance.

Pins in the gage must pass through the holes and proper contact between the datum features and the three datum simulators must be made. This gage will verify that the hole positions relative to one another and to the datum planes are within the .032" tolerance at MMC.

The second segment of the composite tolerance shows a .016" diameter feature-relating tolerance at MMC relative to datum feature A. The gage for this tolerance has only one plate, and it simulates primary datum A. Pins are located in the plate at the true positions of the holes. The pin diameters are sized to the virtual condition created by the .016" diameter position tolerance.

Pins in the gage must pass through the holes and permit the plate to rest flat against datum feature A. This gage checks the relative positions of the holes (the feature-relating tolerance) and also verifies the orientation of the holes relative to datum A.

Position Tolerance on Multiple and Simultaneous Patterns

Holes serve many functional purposes. One very common purpose for holes is to provide clearance for installing the fasteners that hold parts together. In the case of an automobile, there are many holes for attaching the various parts that complete the engine; for example, the pattern of holes for attaching the engine head, or another pattern of holes for mounting the oil pan.

When clearance holes are part of multiple interfaces of unrelated parts, the hole pattern for each of the parts may be separate from all the others. Because there is no direct relationship between the engine head and oil pan mounting positions, the groups of holes used for mounting these parts may be toleranced to act as separate patterns. Tolerances applied to the engine block can clearly show that the two groups of holes act as separate patterns.

In other applications, the fastener holes in a part make up a single interface and act as a single

pattern, even if the holes are of different sizes. Many design applications use dowel pins to establish the location between parts and bolts to clamp the parts together. The holes for the pins are one diameter and the holes for the bolts are a different diameter, yet all the holes need to be related to one another to form a single pattern that will properly assemble. The drawing for such parts should therefore show the requirement for these holes to act as one pattern.

Locations of holes, apparent grouping, and hole size often provide clues as to which holes form a pattern. However, these characteristics can be misleading and are subject to different interpretations. For purposes of position tolerances, the three characteristics listed previously are *not* considered in determining which holes fall into the same pattern. There is a standardized way of defining features to act as a single pattern or to separate them into multiple patterns.

Where position or profile tolerances are applied, features act as a single pattern if the tolerances applied to them reference the same datums in the same order of precedence and with the same material boundary modifiers. This requirement is only applicable to position and profile single-segment feature control frames and the first segment (the pattern-locating tolerance) of a composite position or profile tolerance. It is not applicable to the second or third segment of a composite tolerance.

When it is desired that a specific single-segment position tolerance act as a separate requirement from the other tolerances containing the same datum references, it is necessary to apply a note. The note simply states:

SEPARATE REQUIREMENT

or

SEP REQ

It is also possible to apply the same note to the first segment of a composite tolerance. There is no need to apply this note to the feature-relating segments of a composite tolerance because the feature-relating segments are understood to create separate requirements.

The following paragraphs describe single (simultaneous) and separate patterns. Holes are used in the examples, but the single or separate patterns could be any type of feature.

The designer must determine the functional purpose of the holes and other features in a part and determine whether there is one or more patterns of features. The functions of the features determine how position tolerances are applied on

a drawing, and the position tolerances indicate through the datum feature references and notations whether the features belong to one or more patterns.

Multiple Feature Groups Related to One Datum Reference Frame

Where single-segment feature control frames contain the same datum references in the same order of precedence with the same material condition modifiers, all the toleranced features become a single pattern. The tolerances shown in the feature control frames become a *simultaneous requirement* because they contain identical datum feature references. In effect, the tolerance zone frameworks created by the tolerance specifications are united to become one tolerance zone framework.

Simultaneous Requirement Referenced to Flat Datum Surfaces

Some design applications occur for which all holes need to form a single pattern. See [Figure 9-16](#). The given part has two groups of holes. Each group

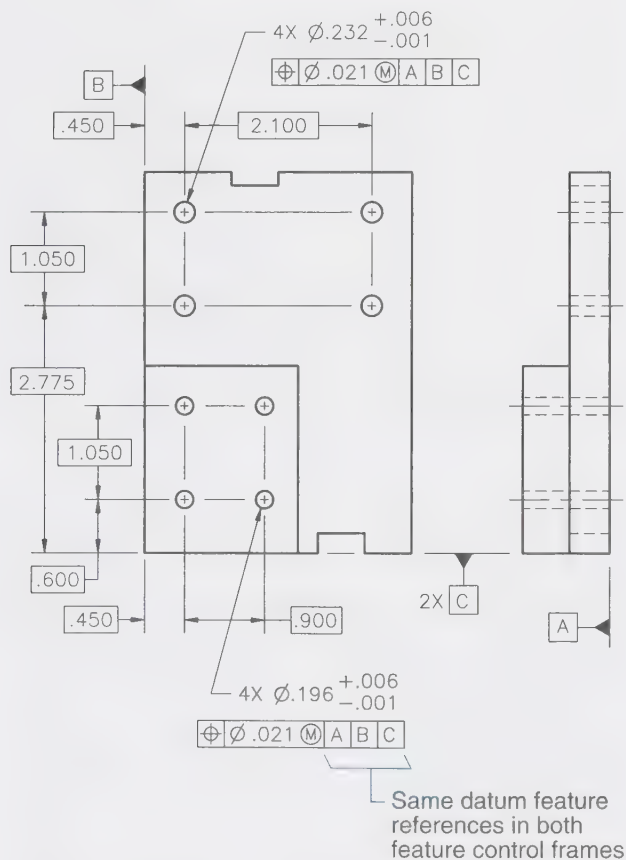


Figure 9-16. The eight holes in the given part create a single pattern because the position tolerances reference the same datums, in the same order of precedence, and with the same material boundary modifiers.

has four holes. The group near the top of the part has four .232" diameter holes. A position tolerance of .021" diameter is referenced to datum feature A primary, B secondary, and C tertiary. The group of holes near the bottom of the part has four .196" diameter holes. A position tolerance of .021" diameter is referenced to datum feature A primary, B secondary, and C tertiary.

Both groups of holes have position tolerances with identical datum feature references; therefore, all eight holes act as one pattern. The given example shows the same tolerance value in both feature control frames. This has no effect on whether the holes act as one or two patterns.

One functional gage may be used to verify the locations of the holes in a single pattern. See [Figure 9-17](#). The shown functional gage will check the eight hole locations of the part in the previous figure.

The shown gage is made of plates that act as datum simulators. Pins located at the true positions are sized at the virtual condition of the holes. A part inserted in the gage must properly contact the three datum simulators with the eight pins passing through the holes.

It is unnecessary to make two gages to check the positions of the holes, although it would not be wrong for this particular part. Because the datum features are flat surfaces, the two gages would each establish identical datum reference frames and result in the same check as a single gage. If two gages were used, they would each have the three datum simulators, with pins located at the true positions of the holes relative to the datums and the pins would be sized to the virtual conditions. Placing the part in each of the two gages provides the same check as placing the part in the single gage that simultaneously checks all eight holes.

Simultaneous Requirement Referenced to Datum Features of Size

Simultaneous requirements are significantly different than separate requirements when the referenced datum features are features of size. See [Figure 9-18](#). The figure shows two groups of holes. Position tolerances for these groups of holes are referenced to the same datums, in the same order of precedence, and with the same material boundary modifiers. Because of the identical datum references, all four holes create one pattern (a simultaneous requirement).

The effect of the simultaneous requirement can be seen by observing the functional gage that can be used for inspection of the hole locations. Only one tool is used to check all holes simultaneously.

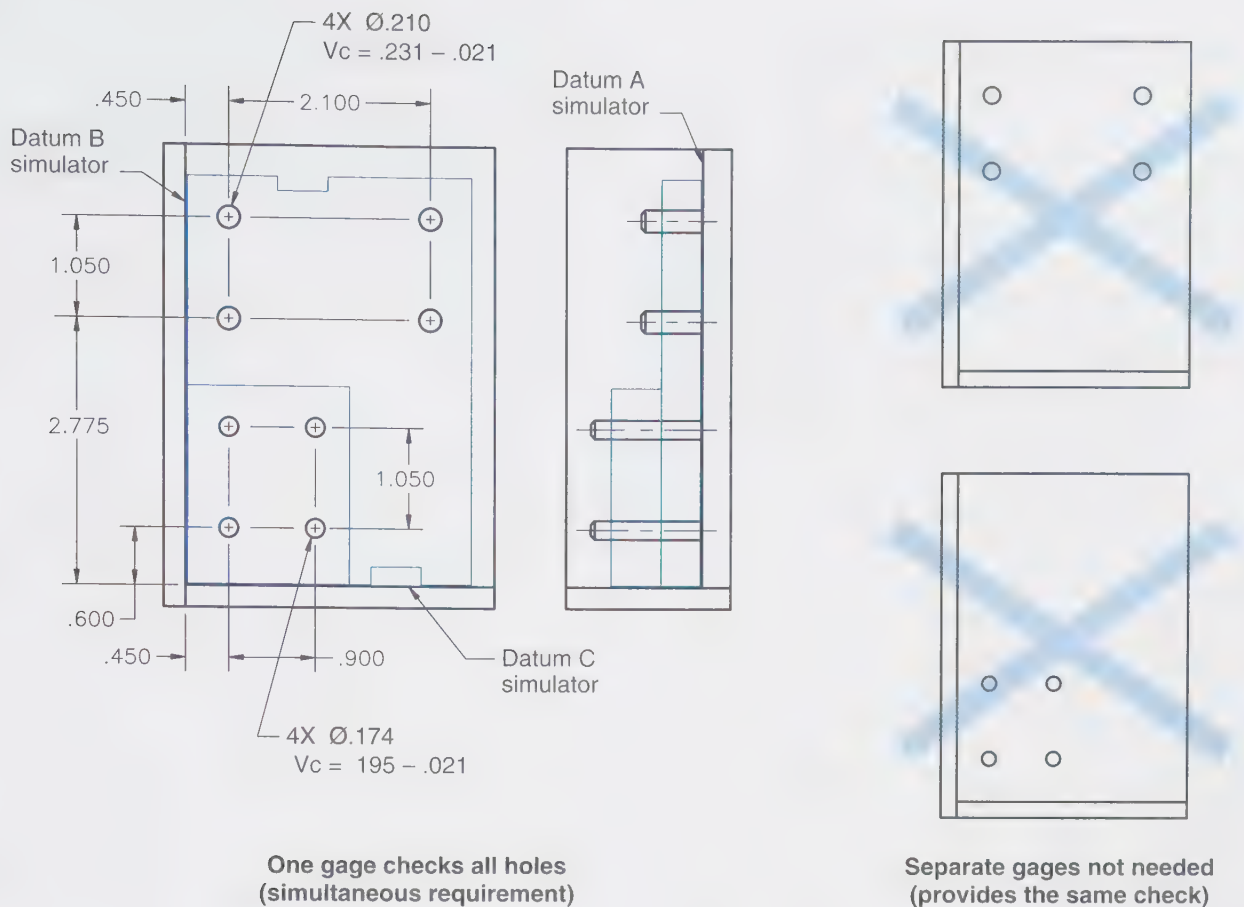


Figure 9-17. Only one gage is required to check the position for all eight holes.

The datum A simulator is sized to the MMC size of primary datum feature A. The datum B simulator is a plate to contact datum feature B. A key on the side of the tool simulates datum C. The key is sized to the virtual condition of the datum feature C slot. Pins for checking hole location are sized to the virtual condition of the holes. These pins are slipped into the tool after the tool is placed inside the workpiece.

If the datum feature C slot is produced at the least material size and at perfect position, the tool is free to translate or rotate on datum axis A by an amount equal to the clearance between the datum feature C slot and the datum C simulator. This allowable movement will permit the tolerance zone framework for the four holes to rotate out of position from the slot, but only by the amount of the clearance between the slot and key. It should be noted that the tolerance zone framework maintains a perfect relationship to the datum reference frame. The relative movement between the datum features and the datum reference frame is what allows the produced holes to move relative to the datum features.

Separate Requirement Noted for References to Datum Features of Size

Groups of features may be specified to allow their tolerance zone frameworks to move independently even though datum references are identical. Exception to the simultaneous requirement is taken by placing the SEPARATE REQUIREMENT notation under the feature control frames of those items that are to act separately. See [Figure 9-19](#). Although this figure contains the same part shown in [Figure 9-18](#), the notation of separate requirement has been added beneath the two position tolerances.

Both sets of holes are separately controlled due to the notation, and each set of holes has a tolerance zone framework that is independent of the other. They are separate. The effect of this type of tolerance requirement is shown by the functional tools for inspection of the two hole patterns. Two tools are shown for checking the two separate requirements. Each tool has datum simulators that are sized in the same manner as described for [Figure 9-18](#). Datum A is simulated at the MMC size of primary datum feature A. The datum B

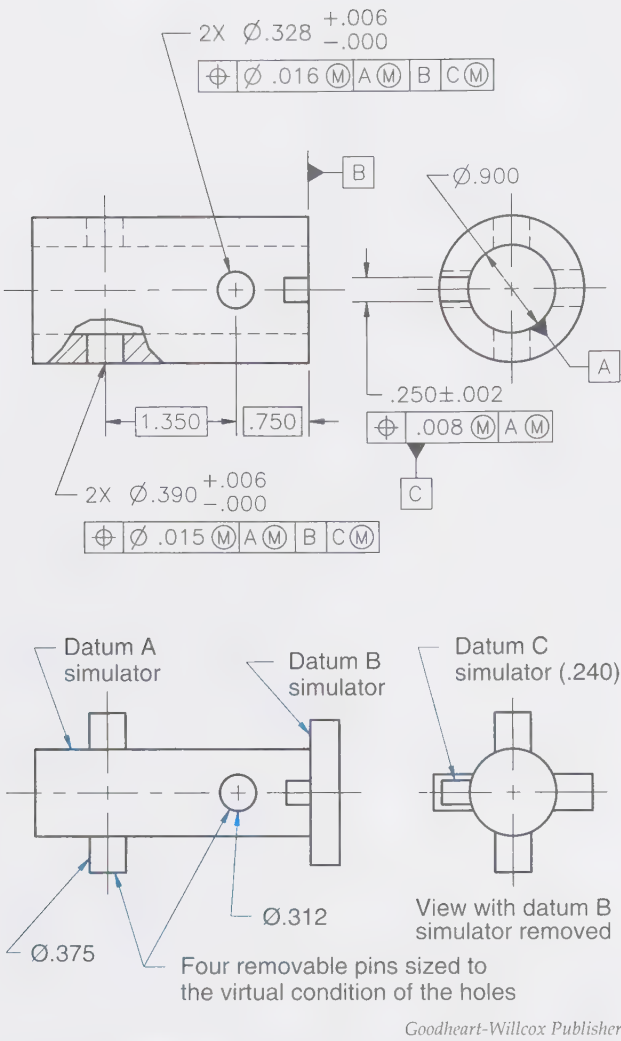


Figure 9-18. Identical references to datum features of size require that the features be considered as a single pattern (simultaneous requirement).

simulator is a plate to contact datum feature B. A key on the side of the tool simulates datum C. Datum C is simulated at its virtual condition because it is tertiary.

The functional gages are free to translate or rotate on datum axis A by an amount equal to the clearance between the datum feature C slot and the datum C simulator key. The two patterns of holes may rotate in opposite directions because their tolerance zone frameworks are not tied together. The separate tools shown in the figure will permit one group of holes to rotate in one direction based on the clearance between the datum feature C slot and the datum C simulator, and the other group of holes may rotate in the opposite direction. The tolerance zone frameworks for both patterns of holes remain perfect relative to their referenced datums. The apparent movement of the holes is the result of the relative movement between the datum reference frame and the datum features that have

departed from their virtual conditions (or maximum material boundaries).

Tolerance specifications should permit the maximum amount of variation that can be accommodated by the function of the part. If the function

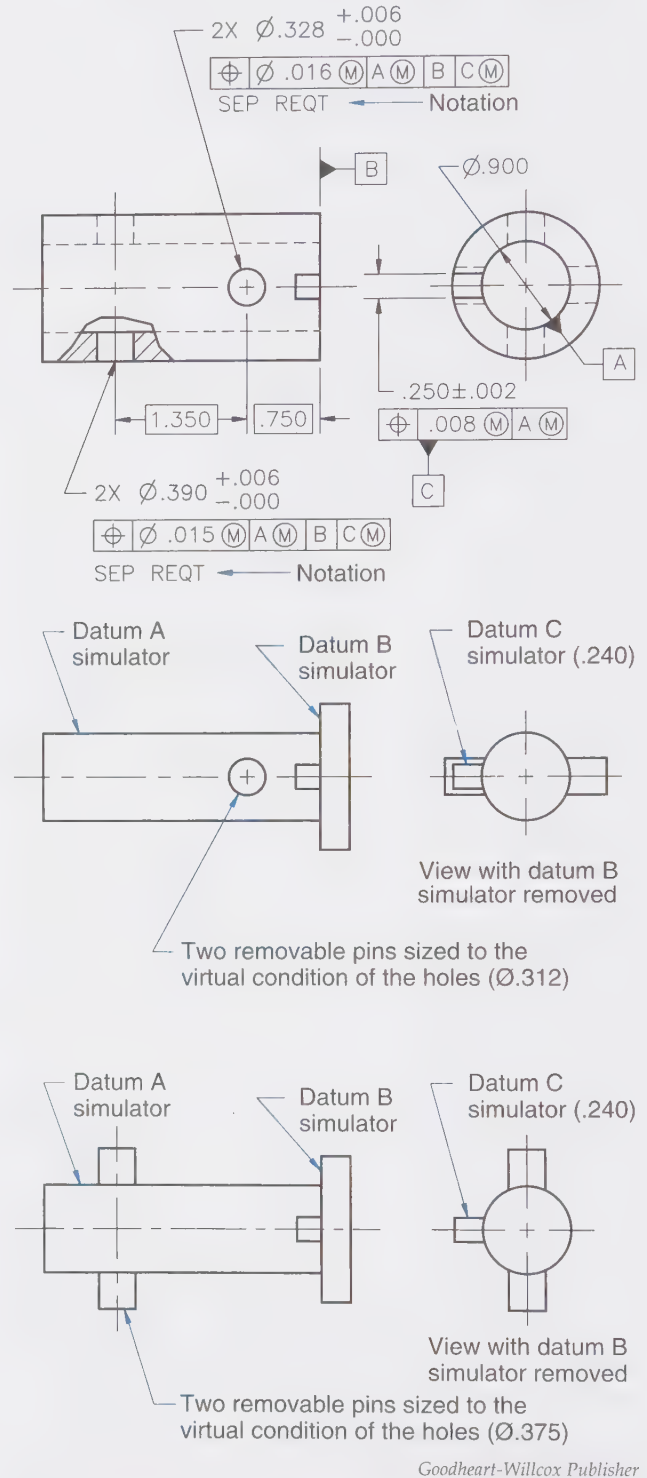


Figure 9-19. A notation of separate requirement under a position tolerance specification requires that the controlled features be verified separate from any other groups of features that are toleranced to the same datums.

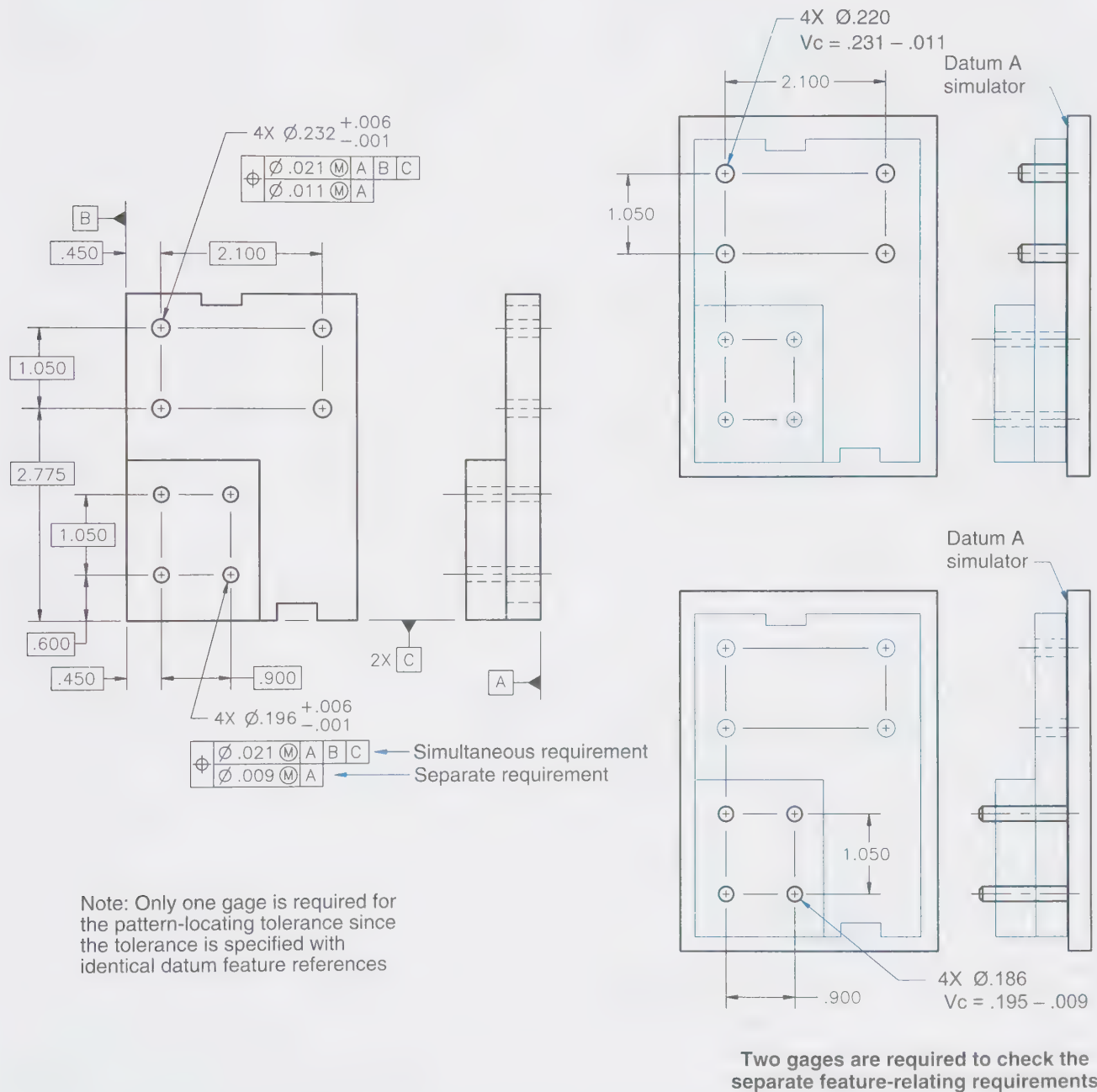
of the part does not require that the holes act as a single pattern, the tolerances should be specified as separate requirements.

Default Simultaneous and Separate Requirements for Composite Tolerances

The first segment of each composite position tolerance specification creates a simultaneous requirement if the datum feature references are identical. However, subsequent segments in composite position tolerances always create

separate requirements unless noted otherwise. See **Figure 9-20**. This figure contains the same part as **Figure 9-16**, but composite position tolerances have been added.

The first segments in the two shown composite position tolerances contain identical datum feature references. The first segment, therefore, creates a simultaneous pattern requirement for the pattern-locating tolerance. The tool shown in **Figure 9-17** can be used to check the hole locations for this requirement.



Goodheart-Willcox Publisher

Figure 9-20. The first segment of multiple composite tolerance specifications creates simultaneous requirements when datum feature references are identical. The second segment creates separate requirements unless noted otherwise.

Based on the current standard, the second segments of the shown composite position tolerances create separate requirements, regardless of the datum references. In effect, the two sets of holes may float in opposite directions within the pattern-locating tolerance. This requires that two functional gages be created to check the two patterns of four holes. The pins in each tool are sized to the virtual condition of the hole that is created by the feature-relating tolerance. Pins in the gages are located at the true positions for the holes.

The separate requirement of the second segment of a composite tolerance is useful when mounting two or more parts that may vary in their relative locations to one another. The relative locations of the parts are still defined by the pattern-locating tolerance, but the feature-relating tolerance establishes only the relative locations of the holes within the pattern.

Noted Simultaneous Requirement for Composite Position Tolerances

Design requirements may require that two sizes of holes be used when mounting a single part to an assembly. Although there are two sizes of holes, the functional requirements make it necessary for all holes to act as a single pattern. If composite position tolerances are applied on two groups of holes that must act as a single pattern, a notation must be added to the composite tolerance specifications. The recommended notation is:

SIMULTANEOUS REQUIREMENT

or

SIM REQ

Placement of the note in line with the affected segment of the composite position tolerance is recommended. See [Figure 9-21](#). Placement under the entire feature control frame might cause confusion because the first segment is already understood to create a simultaneous requirement.

One functional gage may be used to check the second segment of the shown composite position tolerances. The gage is made of one plate with eight pins sized to the virtual condition of the holes. Eight pins in a single plate have fixed locations to one another and simulate one feature-relating tolerance zone framework. Because the pins are sized to their respective virtual conditions, the gage checks the position tolerance that is centered on the true positions created by the feature-relating tolerance zone framework.

It would be incorrect to create a separate gage to check each group of holes. Separately checking the two groups of holes is not acceptable on the

basis of the given tolerance specifications. Checking them separately would permit the groups of holes to float independently, and that is not what the tolerance specifies. The given tolerance specification requires that the holes act as one pattern.

Coaxial Hole Patterns

Coaxial holes are commonly used on parts such as hinges. Coaxial holes often require a closer position relationship to one another (feature-relating tolerance) than to other features on the part. See [Figure 9-22](#). The alignment of coaxial holes must be accurate enough to insert the shaft that passes through them. Location of the holes relative to other features (pattern-locating tolerance) on the part can usually be permitted a larger tolerance.

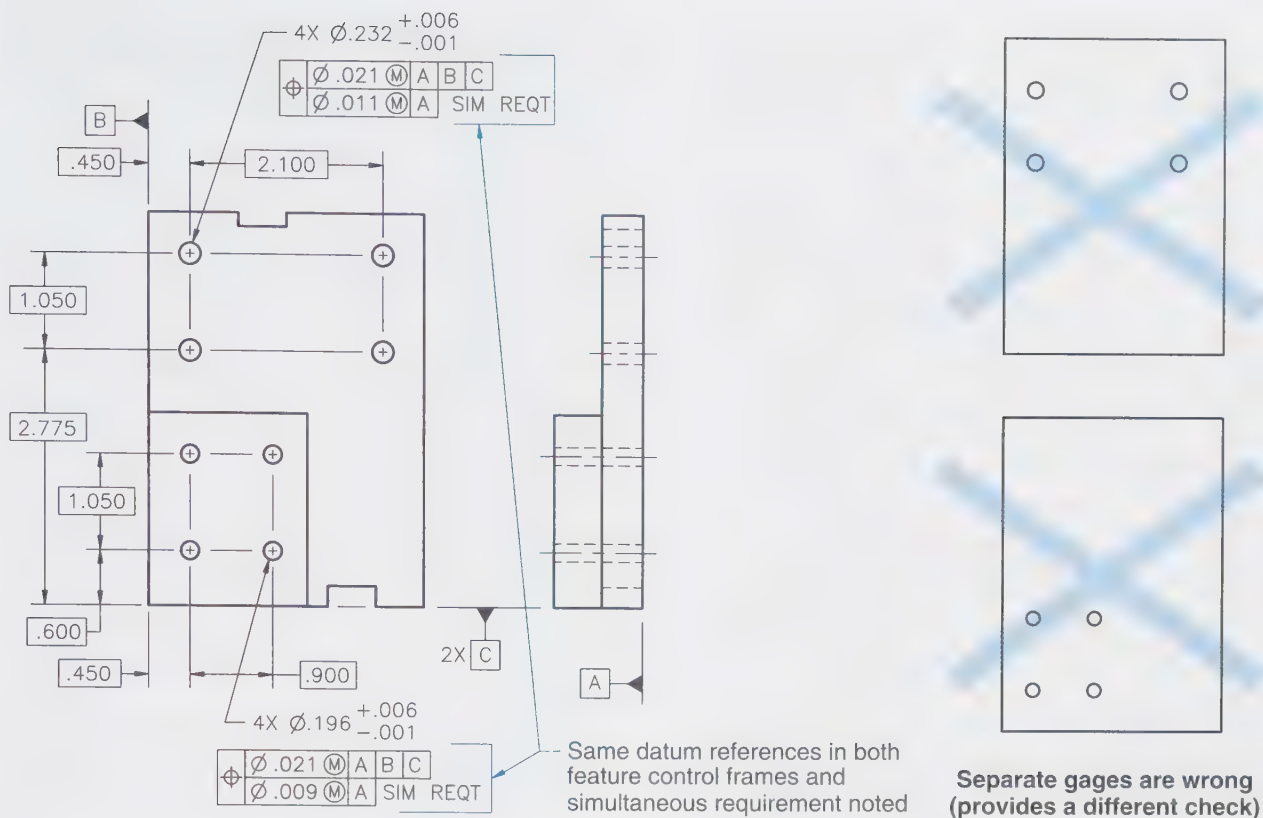
The given figure shows a part containing two coaxial holes. A composite position tolerance is applied to the holes. It shows a pattern-locating tolerance of .016" diameter relative to datum features A, B, and C. This defines the location requirement for the pattern of holes relative to the referenced datums.

The shown feature-relating tolerance is .003" diameter. It is not referenced to any datums. This establishes a .003" diameter tolerance zone for each hole. The two .003" tolerance zones are located on the feature-relating tolerance zone framework, which is a single axis through the coaxial holes. Movement of the feature-relating tolerance zone framework is permitted. It may translate and rotate in any direction because there are no datum feature references. Its position is only restricted by the requirement for the axis of the two holes to be within the .016" diameter pattern-locating tolerance.

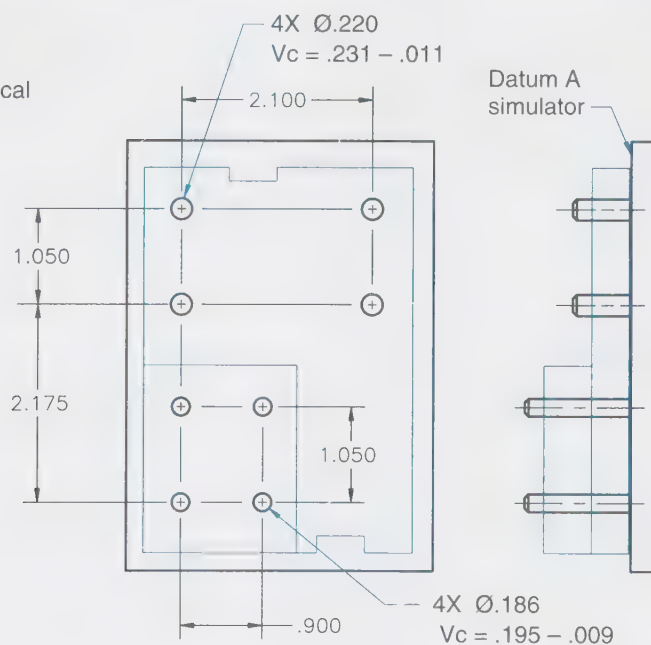
Coaxial holes may have different diameters. See [Figure 9-23](#). A composite tolerance may be applied to coaxial holes of different diameters. To ensure the composite tolerance applies to all holes, the current standard requires that a notation be placed under the tolerance specification. The notation indicates the number of coaxial holes to which the tolerance applies.

The positional accuracy requirements for the axes of the holes are identical to the explanation given for holes of the same diameter. The pattern-locating tolerance locates the holes relative to the referenced datums, and the feature-relating tolerance defines the coaxiality requirements for the holes.

When no datum features are referenced in the feature-relating tolerance, the feature-relating tolerance zone framework may translate and rotate in any direction. The only restriction on the pattern orientation is invoked by the pattern-locating tolerance.



Note: The gage for the pattern-locating tolerance is identical to the one shown in Figure 9-17



One gage checks the feature-relating tolerance on all holes (one simultaneous pattern)

Goodheart-Willcox Publisher

Figure 9-21. When simultaneous requirement is noted on the second segments of multiple composite feature control frames, the feature-relating tolerances create a single feature-relating tolerance zone framework.

It is possible to constrain one or more rotational degrees of freedom of the feature-relating tolerance zone framework. To establish the rotational

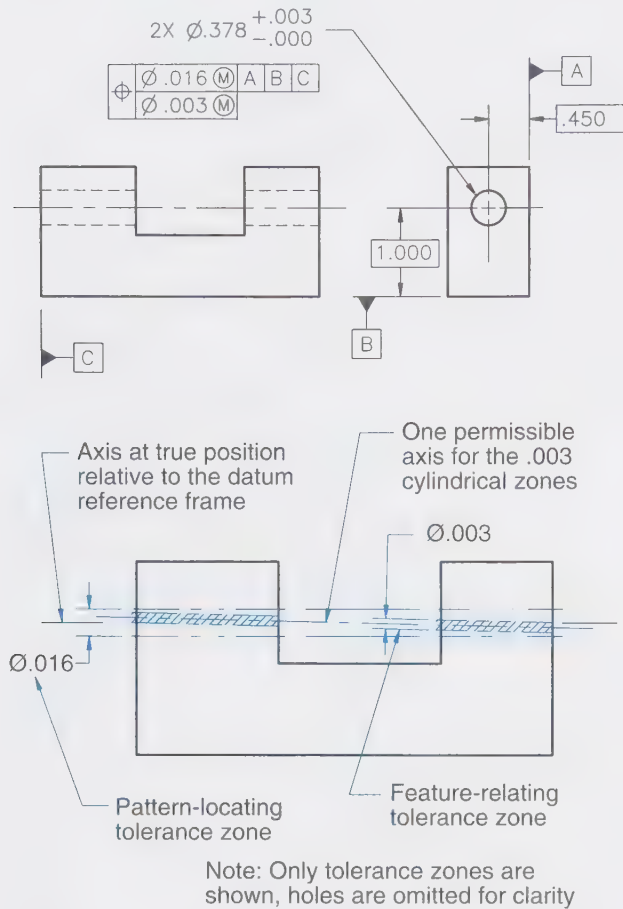
requirement, datum feature references are included in the feature-relating tolerance. See **Figure 9-24**. The number of datum references shown depends on the level of control desired.

References to datum features A and B are made in the feature-relating tolerance of the given figure. The tolerance requires that both holes be located within .006" diameter tolerance zones. Those zones must be located on a feature-relating tolerance zone framework that has rotational degrees of freedom constrained relative to the referenced datums. Proper rotational constraint of the framework for the given figure must result in the framework being parallel to datums A and B.

The reference to datum features A and B in the second segment of the composite tolerance does not invoke any location requirement to those datums. The datum references in the second segment constrain only the rotational degrees of freedom.

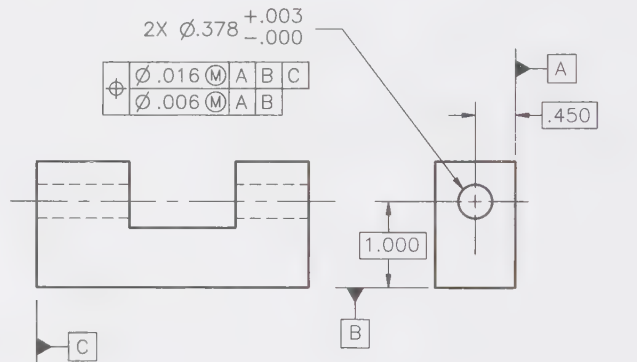
Other Coaxial Tolerances

Two or more coaxial features may be tolerated relative to one another. This is done by



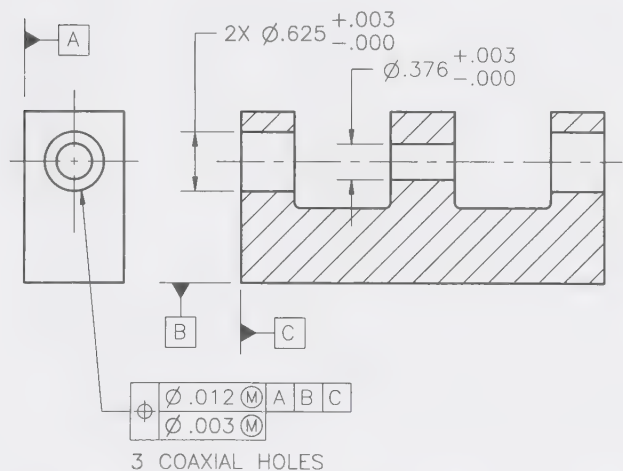
Goodheart-Willcox Publisher

Figure 9-22. A composite tolerance may be applied to coaxial holes.



Goodheart-Willcox Publisher

Figure 9-24. Datum references may be included in the second segment of a composite tolerance that is applied to coaxial holes.



Goodheart-Willcox Publisher

Figure 9-23. When a composite tolerance is applied to coaxial holes of different diameters, a note is placed beneath the feature control frame. The note indicates the number of coaxial holes.

identifying one of the coaxial features as a datum feature and applying a position tolerance on the other features to reference the identified datum. See [Figure 9-25](#).

Position tolerances are typically applied to coaxial features when the main concern is assembly of parts. A properly calculated and applied position tolerance with the MMC modifier will ensure that the parts can be assembled.

The given figure shows a simple stepped shaft. The larger diameter is identified as datum feature A. The other diameter has a position tolerance of .005" diameter at MMC referenced to datum feature A at MMB.

A gage for checking the position tolerance is shown. Primary datum A is simulated at the MMC size. A hole to check the position tolerance on the .567" diameter is sized to the .572" diameter virtual condition. If the stepped shaft fits into the gage, the position tolerance requirement has been met.

Radial Hole Patterns

Radial holes have axes that pass through the center of a circular shape. See [Figure 9-26](#). Four radial holes are shown near the left end of the given part. There is a position tolerance applied to the holes. It specifies a position tolerance of .020" diameter at MMC relative to datum A-B primary and datum C secondary. The two datum references define the required datum reference frame

for the position tolerance applied to the holes. Tolerance zones created by the tolerance specification are located on a framework that is located and oriented relative to the datum reference frame.

The reference to compound datum features A and B establishes an axis through the given part. This axis is the primary datum for the specified tolerance. The tolerance zone framework is centered on and perpendicular to the primary datum axis.

Two of the three planes in the datum reference frame intersect on the primary axis. The third plane of the datum reference frame is perpendicular to the axis and is located relative to datum feature C. Datum feature C does not affect the orientation of the third plane because the plane must be perpendicular to the two planes located on datum axis A-B. It must be perpendicular because the planes of a datum reference frame are always mutually perpendicular.

The tolerance zone framework for the four radial holes is located (two translational degrees of freedom are constrained) and oriented (two rotational degrees of freedom are constrained) relative to the primary datum axis. The framework is located (one translational degree of freedom is constrained) by the secondary datum plane.

Tolerance zones for the four holes are located on the tolerance zone framework. The axis of each hole may lie anywhere within its tolerance zone. There is no requirement for holes on opposite sides of the pattern to lie on a common axis.

Symmetry

Symmetry is a condition where features on each side of a centerline or center plane are equal. Symmetrical features may be hole patterns or any other type of feature.

There are two types of tolerances that may be applied to specify the location accuracy requirements for symmetrical features. They are position and symmetry. The two tolerance types establish significantly different requirements and position should be used to avoid the costly and sometimes confusing requirements of symmetry. Position tolerances are adequate to define symmetrical relationships on an MMC, LMC, or RFS basis. Symmetry tolerances should be avoided because they establish control of a derived feature that is difficult if not impossible to verify.

The 1982 standard did not include the symmetry tolerance symbol. When working to the 1982 standard, all symmetry location requirements must be controlled by position tolerances.

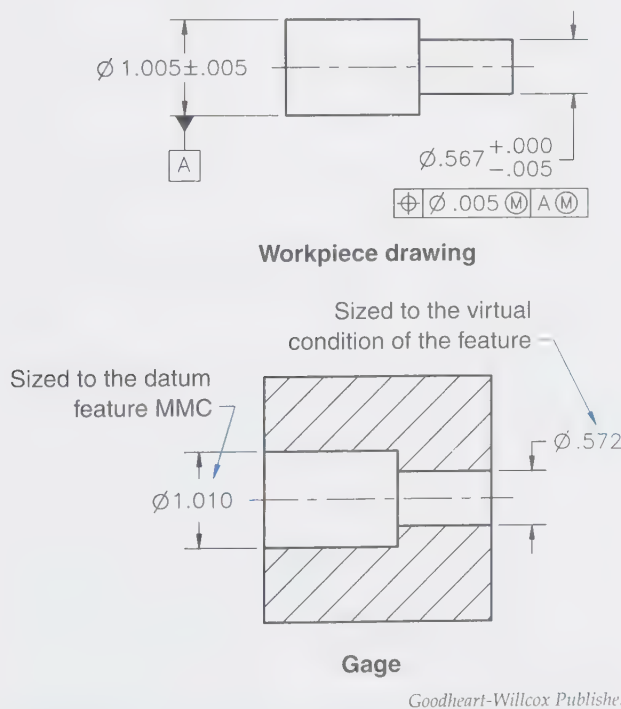
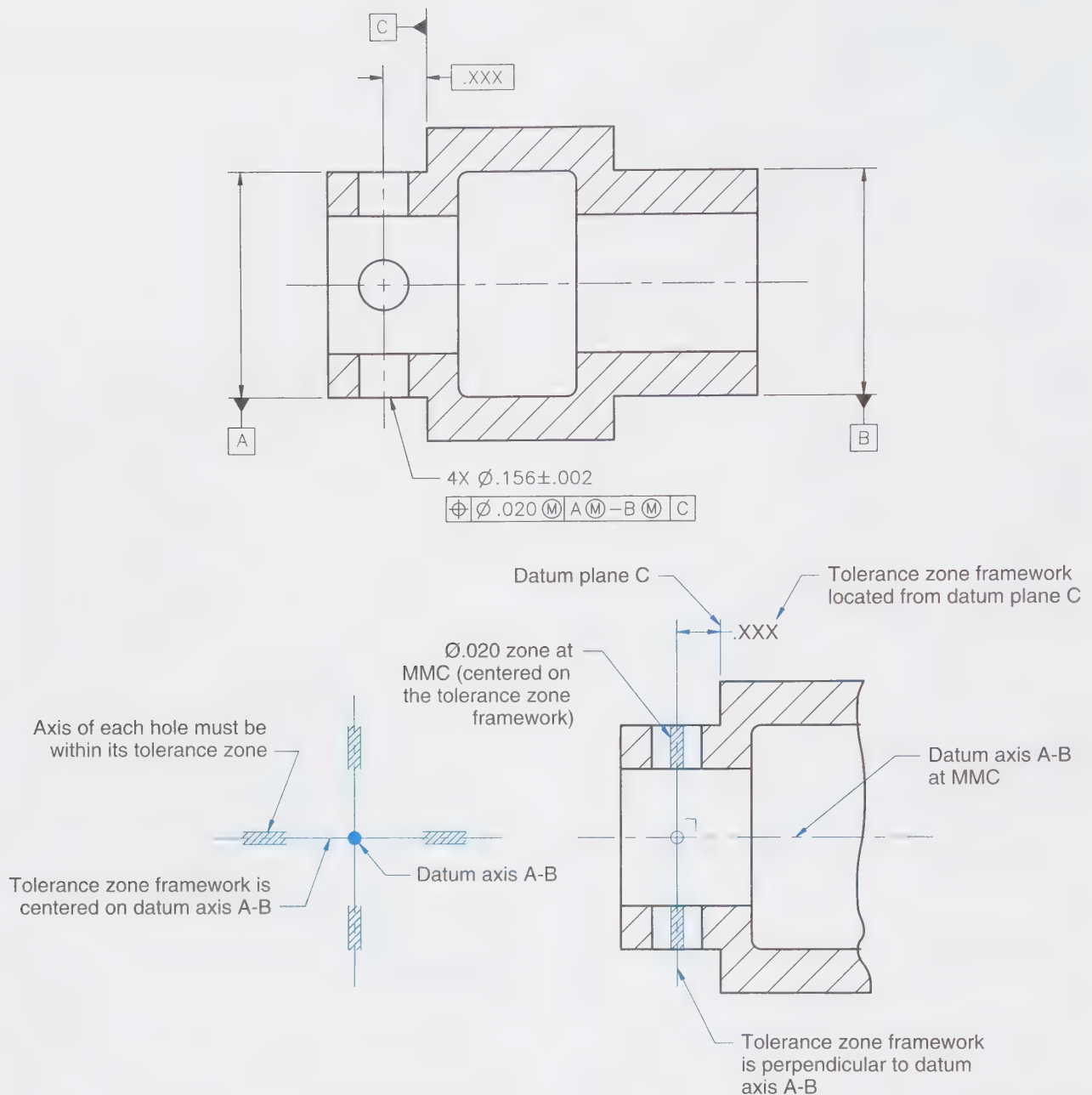


Figure 9-25. Coaxiality of features such as a stepped shaft may be controlled with a position tolerance.



Goodheart-Willcox Publisher

Figure 9-26. Radial holes may move independently within the tolerance zones.

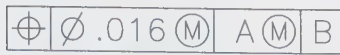
Although the symmetry symbol is in the 1994 and 2009 standard, its meaning is very specific and should be thoroughly understood before using it.

Feature Control Frame

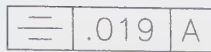
Feature control frames used to indicate symmetrical position requirements are formatted the same as for any other position tolerance specification. See Figure 9-27. Prior to 1982, some symmetrical tolerances were specified using a feature control frame that included a symmetry tolerance symbol. It was perceived that position tolerances could meet most, if not all, symmetry specification

needs, and the symmetry symbol was removed from the 1982 standard in an attempt to reduce errors in tolerance application. There was concern in American industry about the removal of the symmetry symbol from the standard. That concern resulted in the symbol being reinstated in the 1994 standard, and the symbol was given very specific meaning that is different than position tolerances. This difference has continued forward to the current standard.

Position tolerances applied RFS, MMC, or LMC are generally used to define the allowable variation in axis or center plane location. Another



Symmetry of hole patterns at MMC



Symmetry of derived median of features at RFS

Goodheart-Willcox Publisher

Figure 9-27. Two symbols may be used for various symmetry applications, but they have different meanings.

application is for MMC and LMC to create virtual condition boundaries that limit position variation. In addition, only position may be used to specify a requirement of symmetry for a pattern of features. Symmetry tolerances are only used to define the allowable location of the *derived median line* or *derived median plane* created by a single feature, and the need for such a tolerance is rare.

Position for Symmetry of Hole Patterns

One common application of position for symmetry control is on hole patterns. When functional requirements are such that a hole pattern must be located symmetrically to some feature on a part, position tolerances may be applied to express the required control. See [Figure 9-28](#). The position tolerance establishes that the tolerance zone

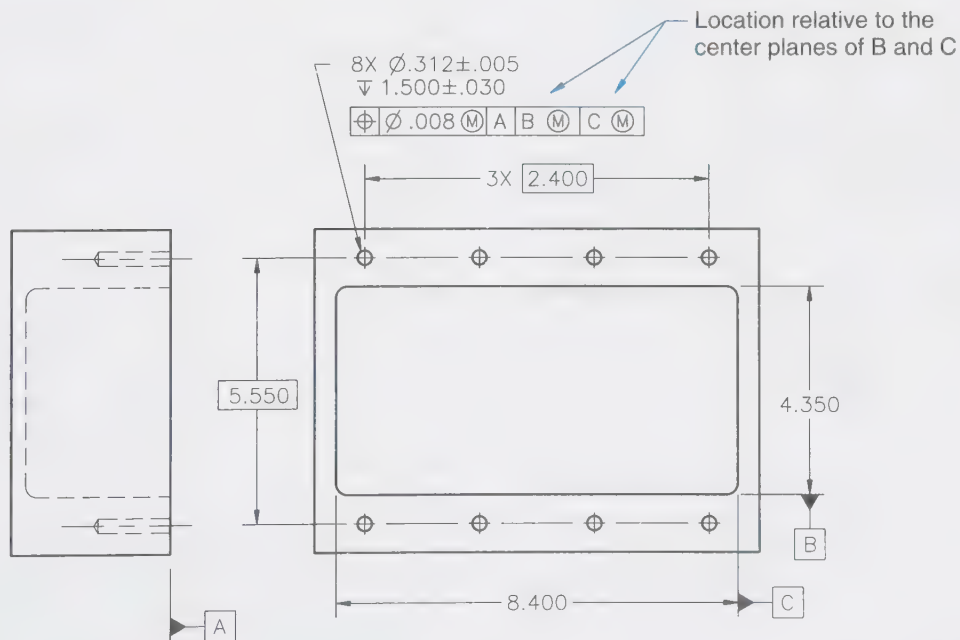
framework for the pattern of holes be located symmetrically relative to the referenced datums.

Dimensions within the hole pattern are applied as they would be for any hole pattern. In the given figure, all holes are dimensioned to show their locations relative to one another.

The location of the hole pattern relative to the datums is defined by the position tolerance. There is no need for dimensions that show the locations of the holes relative to the referenced datums because the pattern of holes is symmetrically located relative to those datums.

The position tolerance specification in the given figure is .008" diameter at MMC relative to datum feature A primary, datum feature B secondary at MMB, and datum feature C tertiary at MMB. Datum features B and C are features of size. These datum features establish center planes.

The position tolerance zones for the eight holes are located on a tolerance zone framework that is symmetrically located relative to the datum planes. The requirement for a symmetrical location is defined by the position tolerance and does not require any dimensions to show the hole locations relative to the center planes. The given drawing does not include any dimensions that show the symmetrical location, and none are required. Attempting to dimension from the datum plane to the plane of symmetry would require a dimension of .000 between the datum plane and the center of the hole pattern. That is not possible to



Goodheart-Willcox Publisher

Figure 9-28. No location dimension from the datum features is required for a hole pattern when position is used to specify symmetry.

show. Applying a dimension from one surface of the datum features would be incorrect. Applying a dimension from the datum plane to one of the holes would result in a basic location that is acceptable, but it is not required.

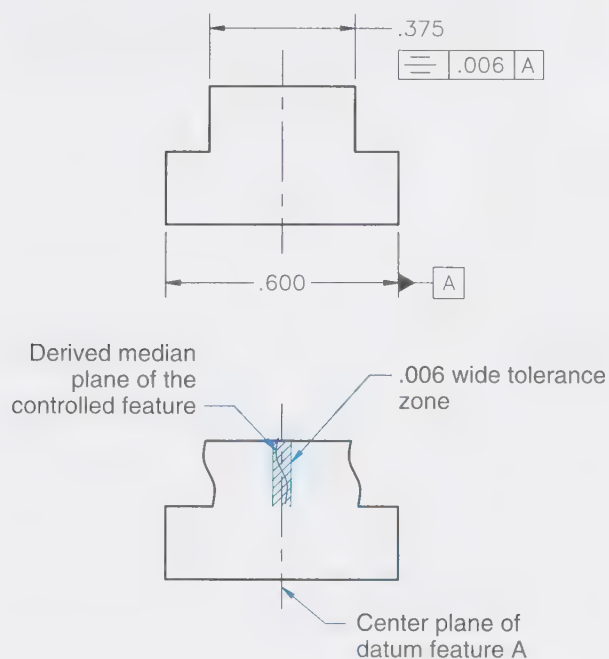
Position for Symmetry on Features of Size

Features of size such as tabs, rails, and slots may be toleranced for location with a positional tolerance or by indicating a symmetry tolerance requirement. An explanation of position tolerances to establish allowable symmetry variation was given in the previous chapter.

Symmetry Tolerance on Features of Size

A symmetry tolerance applied on a feature of size is not the same as a positional tolerance. A symmetry tolerance when applied to a rectangular feature of size requires control of a derived median plane. See [Figure 9-29](#). Dimensions are applied to the features to indicate size. No location dimension is needed when a symmetry tolerance is applied to control the location of the derived median plane.

The given figure shows a rail that is symmetrically located to the base of the part. The base is dimensioned and is identified as datum feature A. The rail is dimensioned and has a symmetry tolerance of .006" at RFS referenced to datum feature A



Goodheart-Willcox Publisher

Figure 9-29. No location dimension is required for a feature of size when a symmetry tolerance is specified for that feature.

at RMB. Symmetry tolerances are always applicable RFS. There is not a dimension that shows the location of the rail.

Datum A is the center plane of datum feature A. The tolerance zone for the rail is .006" wide and is centered on datum plane A, regardless of feature size. The tolerance zone controls the location of the derived median plane (all of the center points) for the toleranced rail. The tolerance zone does not directly control the sides of the rail. Control of the derived median plane only has an indirect effect on the location of the sides of the rail.

It is questionable whether such a tolerance control is ever needed, because control of a derived median plane has little or no functional purpose. The surfaces of the feature are generally the functional features, so when possible, position or profile should be used to establish functional requirements.

Concentricity

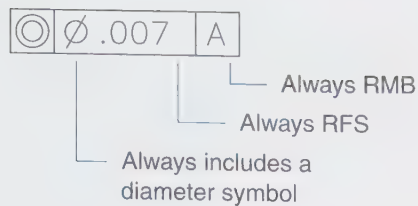
Concentricity requires a derived median line to be located relative to a referenced datum axis. Concentricity is always applied on an RFS basis and the datum reference is always RMB. The use of any other modifier is incorrect. It should be noted that usually, the RFS applications can be met by position, profile, or runout tolerances. It is rare when a concentricity tolerance is needed, and it can be argued that concentricity is never the needed tolerance specification. Where MMC or LMC is applicable, concentricity is not used and the coaxial relationship is specified with position tolerances.

Most coaxial requirements may be met with either a position tolerance at RFS or MMC, depending on design function, profile tolerance, or runout tolerance. These tolerance types can be easily checked on the basis of surface conditions. They are preferable to concentricity tolerances because of the ease in verification, and they can be applied in a manner that meets functional requirements.

There are many applications where concentricity is incorrectly applied, and runout or position is the correct tolerance that should have been applied.

Feature Control Frame

Concentricity tolerances always include a diameter symbol in front of the tolerance value. See [Figure 9-30](#). There must be a datum feature reference. The tolerance must be applied RFS and the datum feature reference must be at RMB.



Goodheart-Willcox Publisher

Figure 9-30. Concentricity always includes a diameter symbol and applies RFS.

Concentricity Tolerance Application

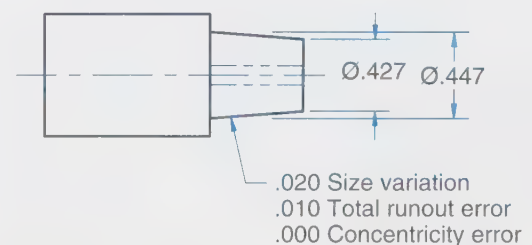
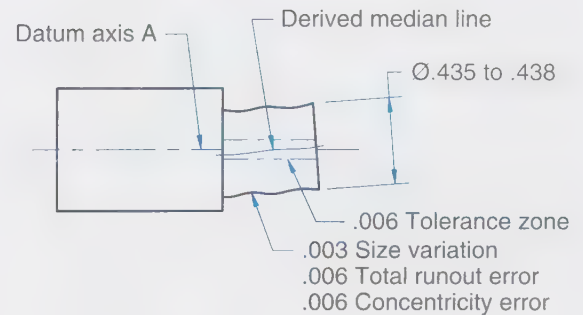
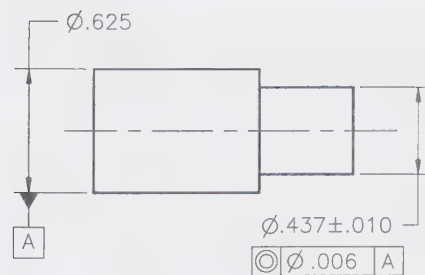
Concentricity may be applied to any feature that creates an axis. **Figure 9-31** shows a concentricity tolerance applied to a cylinder. The tolerance specification in the given figure requires the derived median line of the cylinder to be within a .006" diameter zone that is concentric with datum axis A. The concentricity tolerance does not directly affect the surface condition of the cylinder.

Two possible conditions for parts produced to the given drawing are shown in the figure. The first one shows a cylinder produced at an angle to the datum axis. The size of the cylinder varies between .435" and .438" diameter, which is within the specified limits of size. Measuring the surface of the part results in a full indicator movement of .006", which means there is a total runout variation of .006". *The full indicator movement by itself does not always indicate the amount of concentricity variation.* On this particular part, the shape and orientation of the cylinder does result in a concentricity variation that is equal to the runout variation.

The second produced part shows a tapered feature that ranges from .427" to .447" diameter. This is within the limits of size. The tapered feature is aligned with the datum A axis. Although the tapered feature is aligned with the axis, the full indicator movement across the surface is .010". This indicates a total runout variation of .010". Because the derived median line of the tapered feature is aligned with the datum axis, there is .000" concentricity variation. On this part, the full indicator reading is greater than the concentricity variation. So, concentricity and total runout are not the same.

The two parts show that measuring surface variation with one dial indicator is not adequate to determine the concentricity variation. One of the given parts showed readings that equaled the concentricity variation, and the other part showed readings that far exceeded the concentricity variation.

Verification of concentricity requires that the derived median line location be determined to lie within the allowed tolerance zone. This must be done through a series of measurements such



Goodheart-Willcox Publisher

Figure 9-31. Concentricity establishes a derived median line location, and it can be difficult to evaluate on the basis of surface conditions.

as those that may be taken with coordinate measuring machines and analyzed by computer. The measurements taken must be sufficient to show if the derived median line is within the required boundary. However, it should be understood that defining the exact derived median line location for an object can be very difficult, if not impossible.

It is advisable to specify part requirements through the application of position and runout tolerances when these can meet the functional application for the part. Concentricity tolerances should be avoided whenever possible.

Past Practices

The dimensioning and tolerancing standard continues to evolve as input is received from American industry. Changes to the standard are sometimes necessary to clarify existing guidelines, to expand the guidelines to cover a wider range of applications, and to improve the methods previously created. Methods in the current standard

should be used when working on new drawings unless an existing contract requires compliance with a previous standard.

It is necessary to recognize some of the old practices because many existing drawings were created prior to the current standard. These existing drawings are easier to understand if past practices are known.

Implied Datums

ANSI Y14.5M-1982 made it a requirement that position tolerances include any needed datum references, and did not permit implied datums. Prior to the 1982 standard, the use of implied datums was permitted. **Figure 9-32** shows a position tolerance specification that implies datums. This practice may be encountered if working to parts that were designed prior to 1982.

The hole locations in the given figure are defined with basic dimensions. A position tolerance of .032" diameter is applied to the four holes. No material condition modifier is shown. Because the given drawing is created in compliance with the 1973 standard, MMC is assumed. Prior to the 1982 standard, position tolerances were assumed to apply at MMC. According to the 1982 standard, material condition modifiers must be shown on all position tolerances. Beginning with drawings completed to the 1994 and later standards, all tolerances are assumed to apply RFS.

No datum references are included in the shown position tolerance specification. There are

no identified datum features on the part. Datums must be assumed because they are not identified or referenced.

This part appears relatively simple, and therefore it may seem easy to identify the features that should act as datum features. However, caution should be used because all of the datum features are not obvious.

It is relatively easy to identify two of the datums for this part because basic dimensions for the holes originate at the bottom and left surfaces of the front view. Selecting the third datum feature is not easy.

The mounting surface of a part is typically used as the primary datum. Observing the given drawing, there are three possible planes that could be the mounting surface. One of these three must be assumed to act as the primary datum. After assuming a primary datum, it is necessary to select the order of precedence for the secondary and tertiary datums. Although selection of two of the datum features was easy, determining the optimum order of precedence is left to chance unless the function of the part is known.

Although implied datums were permitted by early versions of the dimensioning and tolerancing standard, they are no longer permitted because of the ambiguity that can result. It is recommended that datum references and datum feature symbols be added to old drawings that include implied datums, especially if those drawings are resulting in parts that do not always function as intended.

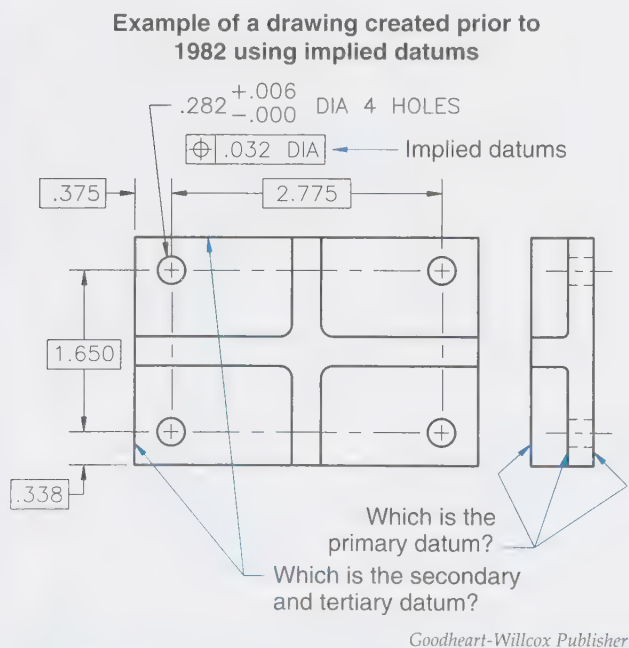


Figure 9-32. Implied datums were permitted prior to the issue of ANSI Y14.5M-1982.

Chapter Summary

- ✓ Composite position tolerances create a pattern-locating and a feature-relating tolerance.
- ✓ The pattern-locating tolerance of a composite position tolerance specification is always larger than the feature-relating tolerance.
- ✓ Paper gaging of the first segment in a composite tolerance is completed in the same manner as for a single-segment position tolerance.
- ✓ The second segment of a composite position tolerance is paper gaged by plotting the relative position variations of the holes and then placing a set of concentric circles over the plotted variations. There is no requirement to center the concentric circles on the origin for the plotted values.
- ✓ A functional gage for the first segment of a composite tolerance is the same as a functional gage for a single-segment position tolerance.

- ✓ A functional gage for the second segment of a composite tolerance must check the relative locations of features in a pattern.
 - ✓ Pattern-locating tolerance zones are located on a pattern-locating tolerance zone framework. This framework is defined by the basic dimensions that locate the pattern relative to datums.
 - ✓ Feature-relating tolerance zones are located on a feature-relating tolerance zone framework. This framework is defined by the basic dimensions that give the relative positions of features in the pattern.
 - ✓ All features that have position or profile tolerances specified with the exact same datum references create a simultaneous requirement (a single pattern). All features in one pattern may be checked with a single functional gage.
 - ✓ Coaxial hole positional requirements may be controlled with a composite position tolerance. The first segment controls hole locations relative to datums and the second segment controls the relative location accuracy for the holes.
 - ✓ Symmetrical patterns of features may be position toleranced to control symmetry relative to specified datums.
 - ✓ Position or runout tolerances should be used instead of concentricity for coaxial features when they provide adequate accuracy to meet functional requirements.
3. The pattern-locating tolerance zones must be constrained in translation and rotation relative to _____.
 - A. the applicable datum reference frame
 - B. the feature-relating tolerance zones
 - C. the production equipment
 - D. None of the above.
 4. A tolerance zone _____ is the set of centerlines that define the true positions on which tolerance zones are located.
 - A. specification
 - B. cylinder
 - C. plane
 - D. framework
 5. Datum references in the second segment of a composite tolerance specification _____ order of precedence than as shown in the first segment.
 - A. may be in a different
 - B. must be in the same
 - C. must be in a different
 - D. None of the above.
 6. Paper gaging to verify the feature-relating tolerance requires that measurements be made _____.
 - A. between features
 - B. from datums
 - C. in polar coordinates
 - D. None of the above.
 7. _____ may be used as datum simulators when the datum features are flat surfaces.
 - A. Flat tool posts
 - B. Spherical-ended tool posts
 - C. Flat plates
 - D. All of the above.
 8. Several groups of holes form _____ if the position tolerances on the holes all have identical datum feature references.
 - A. multiple patterns
 - B. one pattern
 - C. one or more patterns
 - D. a radial pattern
 9. If two groups of holes have position tolerances that are noted as separate requirements, then _____ to check the holes.
 - A. one gage is always adequate
 - B. two gages may be required
 - C. Either A or B.
 - D. Neither A nor B.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. A composite position tolerance has _____ tolerance symbol(s).
 - A. one
 - B. two
 - C. either one or two
 - D. None of the above.
2. Datum references in the first segment of a composite position tolerance constrain _____ of the tolerance zone framework relative to the datum reference frame.
 - A. translation and rotation
 - B. translation
 - C. rotation
 - D. None of the above.

10. If the second segment of a composite position tolerance applied to coaxial holes does not include any datum feature references, then the tolerance controls _____ of the holes.
 - A. only coaxiality
 - B. coaxiality and orientation
 - C. coaxiality and location
 - D. coaxiality, orientation, and location
11. _____ required to define the location of a symmetrical pattern relative to a datum center plane when a position tolerance is used to define the allowable variation of the pattern.
 - A. No dimension is
 - B. One dimension is
 - C. Two dimensions are
 - D. Some dimensions are
12. A primary datum feature reference at RMB requires that the datum simulator be _____ the datum feature.
 - A. in contact with
 - B. fixed in size at the least material limit of
 - C. fixed in size at the maximum material limit of
 - D. fixed in size at the virtual condition of
13. A _____ datum feature reference including an MMB modifier must be simulated at its MMC size if no axis straightness tolerance is applied to the datum feature.
 - A. primary
 - B. secondary
 - C. tertiary
 - D. All of the above.
14. Concentricity tolerance zones control the _____.
 - A. surface of a feature relative to a datum axis
 - B. derived median line of a feature relative to a datum axis
 - C. surface of a feature relative to a datum feature surface
 - D. None of the above.
16. *True or False?* The second segment of a composite tolerance specification is not required to include datum feature references.
17. *True or False?* The pattern-locating and feature-relating tolerance zones may overlap, provided the feature meets all tolerance zone requirements.
18. *True or False?* Hole location variations cannot be plotted on grid paper with sufficient accuracy to determine if composite tolerance requirements are met.
19. *True or False?* Pins in a functional gage for checking hole locations are made equal in size to the MMC size of holes if the holes include a position tolerance of .001" diameter or more.
20. *True or False?* Hole size alone is not a good indicator of which holes form a complete pattern.
21. *True or False?* Two groups of holes may be checked with one gage if the holes act as a single pattern.
22. *True or False?* Coaxiality of features, such as the cylinders on a step shaft, may be controlled with a position tolerance.
23. *True or False?* Symmetry is when a feature or group of features are dimensioned with a nominal offset to one side of a centerline or center plane.
24. *True or False?* Concentricity can be perfect when the feature surfaces include large total runout (surface) variations.
25. *True or False?* Implied datums are permitted on position tolerances.
26. *True or False?* The datum simulator for a secondary datum reference at RMB must force the workpiece into alignment with the simulator, even if this pulls the workpiece off the primary datum simulator.

True/False

15. *True or False?* A composite position tolerance requires a pattern-locating tolerance that is smaller than the feature-relating tolerance.

Fill in the Blank

27. The _____ segment of a composite position tolerance specifies the feature-to-feature location requirements.

28. When paper gaging, _____ circles are used to represent tolerance zone diameters.
29. A functional gage for checking hole locations should include pins that are sized to the _____ of the holes.
30. A notation that states _____ may be placed under the feature control frame if it is necessary to have a group of features act as a separate pattern.
31. Coaxial hole patterns on parts such as hinges have holes that lie on a common _____.
32. Drawings completed in compliance with the _____ issue of the standard shall not include the symmetry symbol.
33. A symmetry tolerance used to locate a tab, rail, slot, or other rectangular feature actually creates a tolerance zone that contains the _____ instead of the feature surfaces.
34. Concentricity is difficult to verify because the _____ of the controlled feature must be derived from surface variations.
35. A secondary or tertiary datum feature of size that is referenced at MMB requires the datum simulator be sized to the applicable _____ of the datum feature.

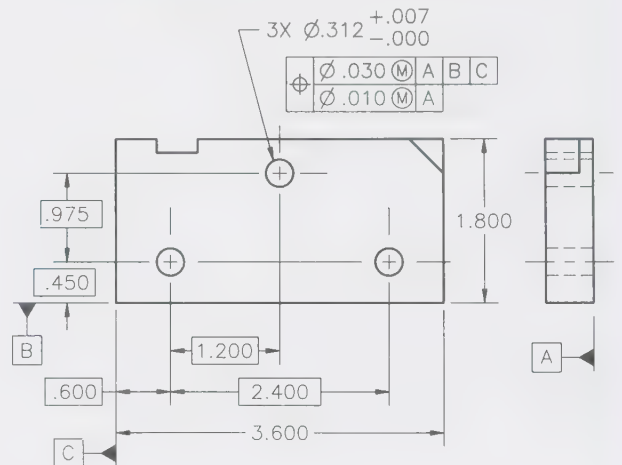
Short Answer

36. Two tolerance zone frameworks are created by a composite position tolerance. What is the name of the framework created by the first segment of a composite position tolerance?
37. Describe the requirements of the first segment in a composite position tolerance.
38. What is the effect of referencing one or more datum features in the second segment of a composite position tolerance that is applied to coaxial holes?
39. Describe a problem that makes it unacceptable to use implied datums.
40. How is a datum feature of size simulated if it is referenced as primary, the datum reference includes an MMB modifier, and the datum feature has no form tolerance applied?

Application Problems

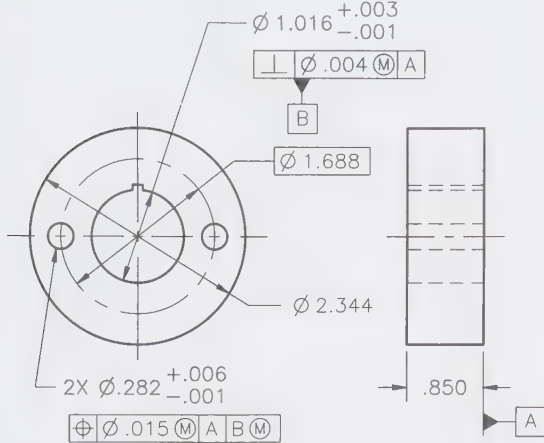
Some of the following problems require that a sketch be made. All sketches should be neat and accurate. Some problem descriptions require the addition of dimensions for completion of the problem. Apply all required dimensions in compliance with dimensioning and tolerancing requirements. Show any required calculations.

41. Draw a composite position tolerance feature control frame that requires a .027" diameter pattern-locating tolerance that is related to primary datum feature A, secondary datum feature D, and tertiary datum feature F. It must require a .012" diameter feature-relating tolerance that is related to primary datum feature A.
42. Make a sketch that shows the pattern-locating tolerance zones for the given drawing. Show datums, true position requirements, and tolerance zones. Do not show the holes. Superimpose feature-relating tolerance zones. The feature-relating tolerance zones must be shown with some amount of permissible offset from the true positions for the pattern-locating tolerances. See **Figure 9-4**. Include dimensions and notes to clarify requirements.

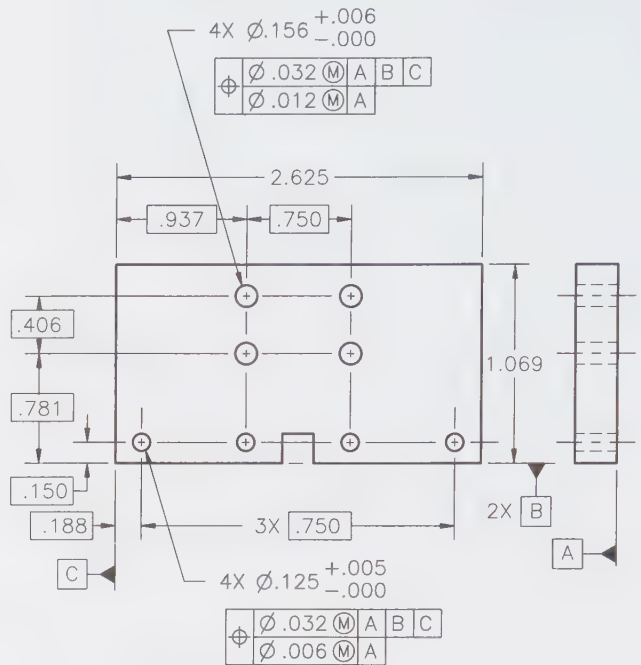


43. Sketch a functional gage that checks hole locations for the pattern-locating tolerance in problem 42. Include dimensions, but assume that a gage needs no tolerance.
44. Sketch a functional gage that checks hole locations for the feature-relating tolerance in problem 42. Include dimensions, but assume that a gage needs no tolerance.

45. Sketch a functional gage for the given drawing. Include dimensions, but assume that a gage needs no tolerance.



46. Sketch the necessary functional gages to check the feature-relating tolerances in the given drawing. Include dimensions, but assume that a gage needs no tolerance.



Chapter 10

Runout

Objectives

Information in this chapter will enable you to:

- ▼ Describe the two types of runout tolerances.
- ▼ Complete an interpretation drawing showing how each of the runout tolerances is measured.
- ▼ Apply both types of runout tolerances on circular features and face surfaces.
- ▼ Specify runout tolerances using multiple datum feature references.
- ▼ Limit the area of application for a runout tolerance.

Technical Terms

circular runout
multiple datum features
runout
total runout

Introduction

Runout is the surface variation that occurs relative to an axis of rotation. These variations may occur on a cylindrical surface that is parallel to the axis of rotation, on a face surface that is perpendicular to the axis of rotation, and in some cases on other surfaces that have circular elements.

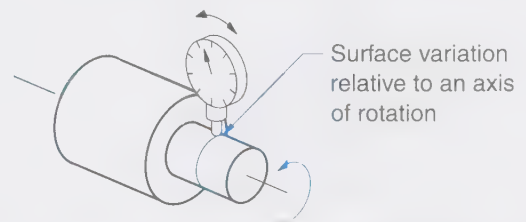
Runout tolerance specifications define the amount of runout variation that is permitted on a surface. The tolerance symbol in the feature control frame indicates whether the permitted variations are circular or total runout. Both types of runout tolerance specifications must always include one or more datum feature references.

Circular Runout

Circular runout is the surface variation at each circular element on a part. The allowable form and location of the circular elements relative to a datum axis are defined by a circular runout tolerance. So, the roundness of a cylinder is controlled and its location on a datum axis is controlled. When circular runout is applied on a surface that is perpendicular to the datum axis, the orientation of the circular elements is controlled. So, the circular runout will prevent wobble of the face surface on a rotating part. The circular runout tolerance value is not limited by the size limits, and size limits are not affected by a circular runout tolerance.

The amount of circular runout at each circular element is evaluated separately from all others. The surface variations are measured relative to an axis of rotation. See **Figure 10-1**. The given part is a simple step shaft.

The circular runout of a single circular element on the small diameter cylinder can be measured by placing a dial indicator in a fixed position and rotating the part on an axis of rotation created by the large diameter cylinder. The full indicator



Goodheart-Willcox Publisher

Figure 10-1. Individual circular elements are measured relative to an axis of rotation to verify circular runout tolerances.

movement caused by one complete revolution of the part is the runout variation at the measured location.

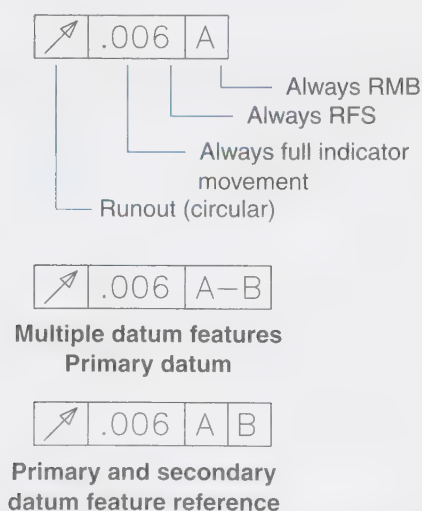
Multiple circular elements on the feature must be measured when the surface is large enough to take multiple measurements, and each measurement is evaluated separately from all others. The largest reading obtained for all the measured circular elements on the toleranced feature is the circular runout variation for the part. This worst-case measurement must be less than the specified circular runout tolerance. If the circular runout variation at any circular element exceeds the specified tolerance, the measured feature is not within the allowable tolerance.

Feature Control Frame

The circular runout tolerance specification has a required format. See **Figure 10-2**. The circular runout tolerance symbol is shown first. It is a single arrow. The arrowhead may be filled or unfilled. The tolerance value follows the symbol. A diameter symbol is never used because the runout tolerance indicates surface variations. A diameter symbol would incorrectly indicate a cylindrical tolerance zone.

Runout tolerances are always applied regardless of feature size. The tolerances must be specified at RFS because the measurements are made on the surface of the part.

An axis of rotation must be established by a reference to one or more datum features. A single datum feature reference to a feature of size, such as a cylinder, is often adequate to establish an axis. It is sometimes necessary to use more than one feature.



Goodheart-Willcox Publisher

Figure 10-2. Runout tolerances are always specified at RFS and must include one or more datum feature references.

The number of datum feature references and selection of datum features is usually determined from how the part is supported in its assembly. If one feature supports the part, then only one datum feature reference is needed. If two bearing surfaces support the part, then multiple datum features (sometimes called compound datum features) are referenced with a dash separating the datum feature letters. If a primary bearing surface and a secondary surface support the part, then primary and secondary datum features are referenced.

It is not acceptable to reference datum features within a runout tolerance in a way that establishes more than one axis of rotation. For runout tolerances, a part may only be rotated on one axis at a time.

Circular Runout Applications

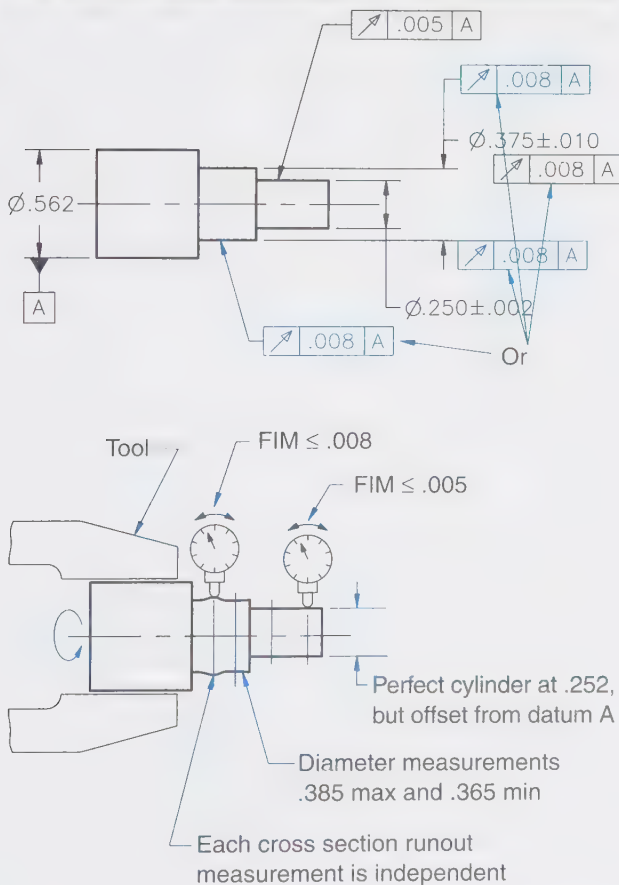
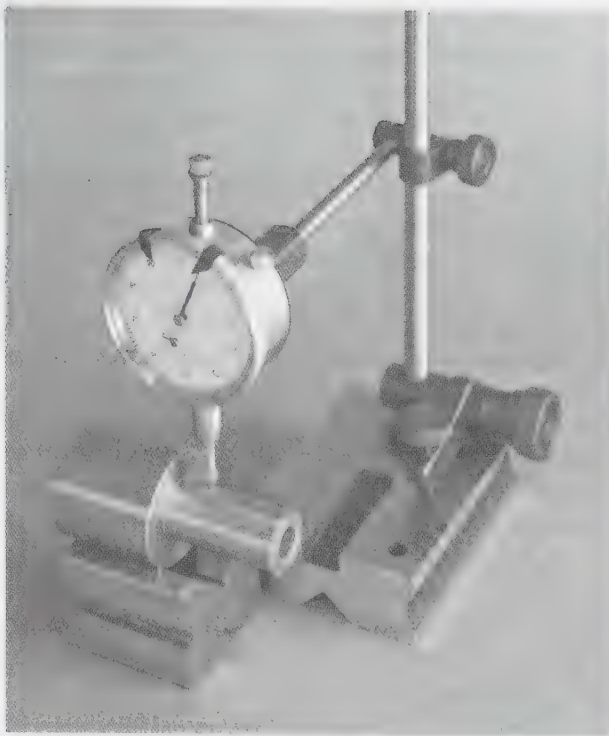
Surfaces with circular cross sections may be controlled relative to an axis of rotation by using a circular runout tolerance. Typical surfaces that include circular elements are cylinders, cones, spherical surfaces, and flat surfaces.

Circular Runout Applied to a Cylinder

The feature control frame for circular runout may be applied in any of three ways. All three indicate the same requirement on the indicated feature. See **Figure 10-3**. The given step shaft has three cylindrical segments. The .562" diameter is identified as datum feature A. Circular runout tolerances on the shaft reference this datum feature to establish an axis of rotation.

Multiple methods of specifying a circular runout tolerance are shown. The feature control frame may be applied to an extension line, connected to the surface by a leader, or applied to the size dimension. Each of the methods has exactly the same interpretation. Any of the shown application methods may be used when completing a drawing. However, application on the size dimension value tends to cause confusion and is not a recommended practice. Industry has made requests that this method be removed from the ASME standards. If for some reason it is desired to apply the runout tolerance on the size dimension, that may be accomplished by placing the feature control frame adjacent to the size dimension or by attaching it to an extension of the dimension line.

Circular runout on a part produced to the given drawing may be checked through the following procedure. Datum feature A must be used to establish an axis of rotation. This can be accomplished by clamping the feature in a collet or chuck. Of course, it is assumed the collet or chuck



Goodheart-Willcox Publisher

Figure 10-3. The datum feature reference in a runout tolerance determines which feature is used to establish an axis of rotation for verification of the tolerance.

has accuracy adequate for use in measuring the specified tolerance.

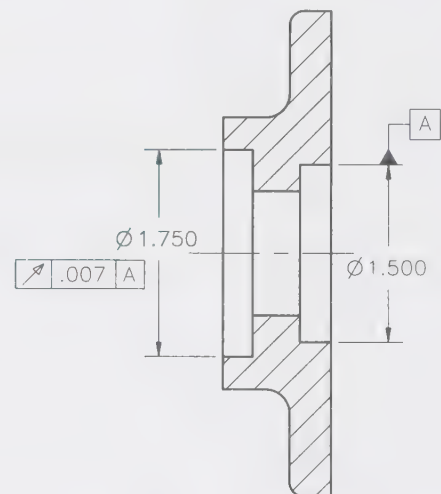
Multiple, but separate, dial indicator readings must be taken along the length of each toleranced feature. Each check must be made with the dial indicator in a fixed location, and the part must be rotated one full revolution. Provided the full indicator movement for each circular check is equal to or less than the specified circular runout tolerance, the feature is acceptable.

The full indicator movement at each cross section on the .375" diameter of the shown part must be less than or equal to .008". The circular runout variations on each cross section of the .250" diameter must be equal to or less than .005".

Applications of runout tolerance are most commonly seen on cylindrical features or on surfaces perpendicular to the axis of rotation. See **Figure 10-4**. Cylindrical features may be internal features such as holes or counterbores. Regardless of whether the runout tolerance is applied to an internal or external feature, the tolerance is a surface control relative to an axis of rotation. The tolerance zones apply along the full extent of the feature.

Circular Runout Applied to Noncylindrical Features

Circular runout may be applied to noncylindrical features if the features include circular elements. The feature control frame is applied using a leader pointing to the surface. It may also be attached to an extension line from the surface.



Goodheart-Willcox Publisher

Figure 10-4. Runout tolerances may be applied to internal features, and they may also reference internal datum features.

It must not be applied to a dimension value. See [Figure 10-5](#). A circular runout tolerance of .010" is applied to a conical surface. This requires that each circular element along the length of the cone be controlled. Surface variations on the cone are measured with the dial indicator positioned normal (perpendicular) to the cone surface.

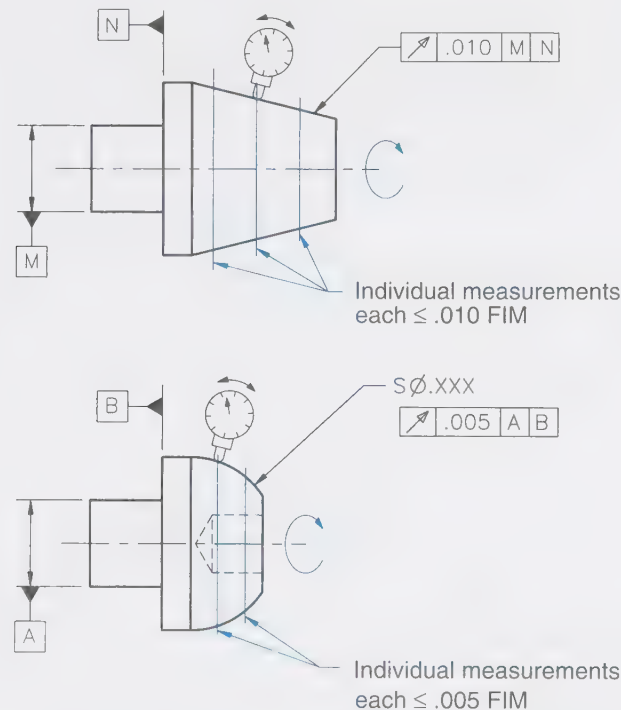
The given figure shows a circular runout tolerance of .005" applied to a spherical surface. Each circular element perpendicular to the datum axis must be controlled to within the .005" tolerance. The runout variations must be measured with the dial indicator positioned normal to the surface.

Circular Runout Applied to Face Surfaces

Circular elements can exist on the face surfaces of an external or internal feature. The face surface may be a flat surface, a large-angle conical surface, or a large-radius spherical surface. See the examples in [Figure 10-6](#).

Circular runout tolerances may be applied on extension lines from these surfaces or connected to the surface with a leader. Both application methods result in the same tolerance requirement. A circular runout tolerance must not be applied to a dimension value when the tolerance applies to a flat surface.

The given figure shows a .008" circular runout tolerance applied to a flat face surface. This



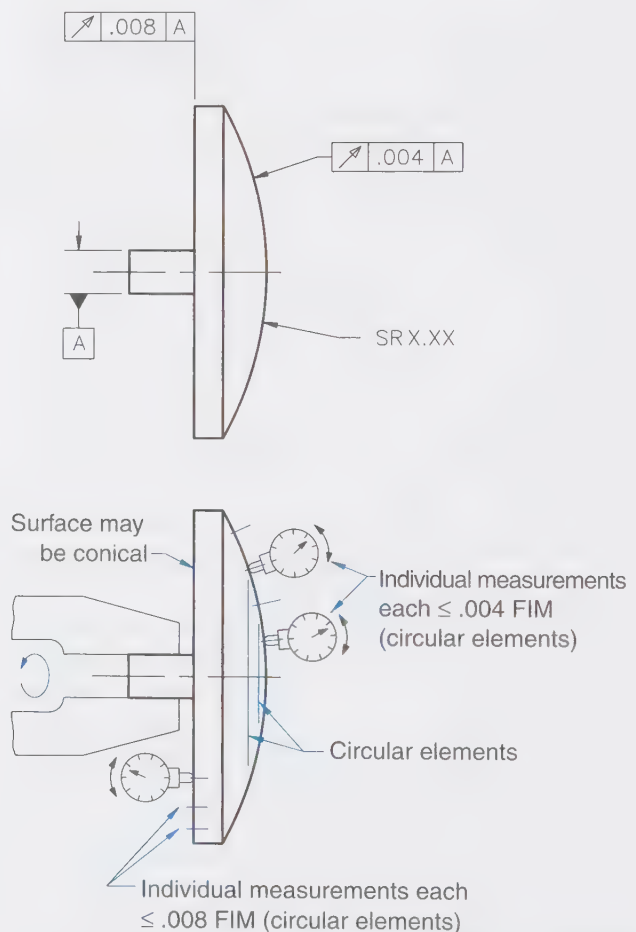
Goodheart-Willcox Publisher

Figure 10-5. Circular runout may be specified on any feature having circular elements that may be related to an axis of rotation.

requires that each circular element on the surface be flat and properly oriented within the .008" tolerance when the part is rotated on the datum axis. This is checked by positioning a dial indicator perpendicular to the surface and rotating the part. If the full indicator movement is less than or equal to .008", then the circular element is acceptable. Multiple circular elements must be checked when possible.

Each circular element on the surface is checked separately. This would permit the surface to be wavy or conical and still meet the requirement for each circular element to be within the runout tolerance. Circular runout applied to a face surface does not control the overall shape of the surface; it only controls individual elements.

A circular runout tolerance of .004" is applied to a large radius spherical face on the given part. Measurements of the circular elements are completed with the dial indicator positioned normal to the surface.



Goodheart-Willcox Publisher

Figure 10-6. Flat, conical, and spherical face surface variations may be controlled with circular runout tolerances.

Circular runout applied to a face surface controls the amount of allowable wobble that may be present when the part rotates. It is not meant to control the shape of the overall surface.

Datum Axis

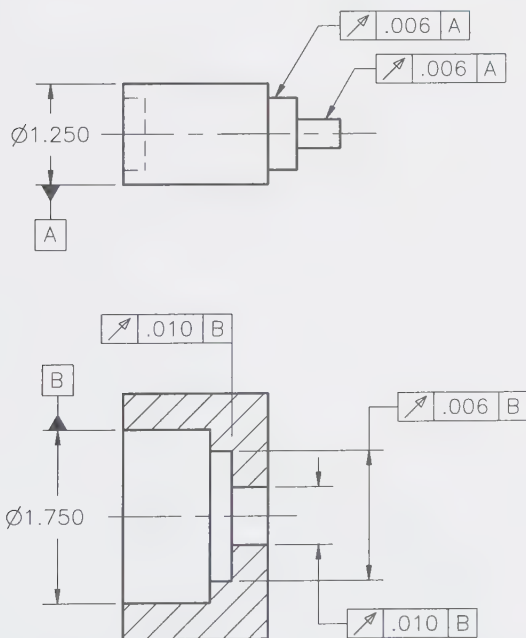
All runout tolerances must include one or more datum feature references that may be used to establish an axis of rotation. Datum features may be identified as entire surfaces or by datum targets.

Datum feature selection must be completed on the basis of design function, with consideration given to the manufacturing and inspection process. It is not very practical to specify tolerances that cannot be verified by available manufacturing and inspection equipment.

Some of the factors that affect whether or not a tolerance can be produced or verified are size, proportions, and relative locations of controlled features and the referenced datum features. The size and relative proportions of features must be considered when specifying runout tolerances because these factors affect manufacturing processes. In addition to size and relative proportions, relative locations should also be considered.

One Datum Feature

Reference to one datum feature is often enough to establish an axis of rotation. See [Figure 10-7](#). The



Goodheart-Willcox Publisher

Figure 10-7. The size and proportions of features should be considered when assigning runout tolerances and identifying datum features.

step shaft has only one datum feature; it is identified as datum feature A. All circular runout tolerances for the shaft reference datum feature A.

Datum feature A is large enough to permit establishing a reliable and repeatable location for the axis of rotation, in this case the datum axis. When the part is produced, a chuck or collet can clamp on datum feature A while cutting the two small diameters. The part will be stable in the manufacturing setup, and the forces applied while cutting the small diameters will not move the part within the chuck or collet.

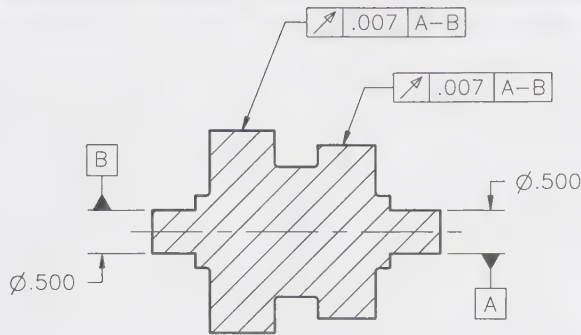
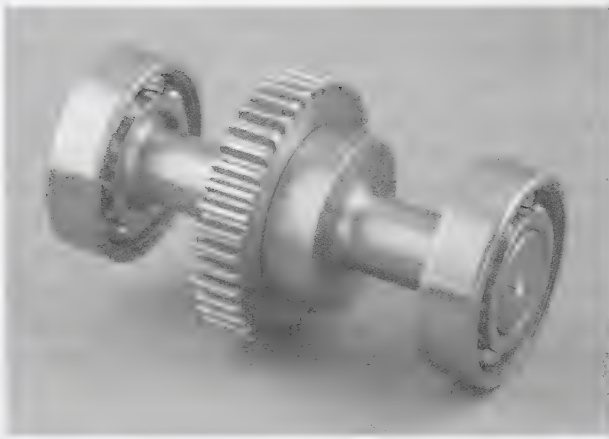
Improper selection of a datum feature on this part could make it difficult to produce and inspect. Selecting the wrong feature to establish the datum axis may increase product cost or even make the tolerances impossible to verify. Consider selection of the smallest diameter as datum feature A. The datum would be difficult to establish accurately because of the large size of the part relative to the small diameter. The mass of the part alone would tend to cause the part to move in the chuck or collet.

Holes, counterbores, and countersinks are examples of internal features that may be used to establish a datum axis. The given figure shows a counterbore that is identified as datum feature B. On this particular part, the counterbore exists for installation of a bearing. Because the bearing establishes the axis of rotation for the part, it is logical to use the counterbore to establish the datum axis. The size of the counterbore relative to the size of the part is also large enough to support the part during fabrication and inspection.

Multiple Datum Features

Many rotating parts are supported by two bearings. Because two features are used to establish the axis of rotation of the part, the drawing should identify those features to establish the datum axis. Two datum features used to establish a single datum are identified as **multiple datum features** in the current standard. Care must be taken not to confuse the term multiple datum features used to establish a single datum with the possible use of the phrase multiple datum features meaning those referenced as primary and secondary. In the past, the name *compound datum features* has been used for datum feature references that utilize more than one datum feature to establish a datum.

Multiple datum features referenced with a hyphen between the datum feature letters in a feature control frame indicate one datum. See [Figure 10-8](#). Two or more datum letters are placed in a single cell within the feature control frame. The letters must be separated by a hyphen. The



Goodheart-Willcox Publisher

Figure 10-8. Two diameters are shown with runout tolerances referencing a datum established from multiple datum features.

two letters identify the two datum features that establish the datum.

The given figure shows a small shaft. Bearing surfaces at each end of the shaft are identified as datum features. One is identified as datum feature A, and the other is identified as datum feature B.

When installed in the functioning assembly, the two datum features are inserted into bearings. The axis of rotation is therefore established by both of these features. To establish a datum axis that simulates the functional application of the part, both datum features must be used simultaneously to establish the datum axis.

The runout tolerances in the given figure reference multiple datum features A and B. This is done by showing datum letters A and B in a single cell. The two letters are separated by a hyphen. This indicates that the part must be rotated using both diameters to establish a single datum axis.

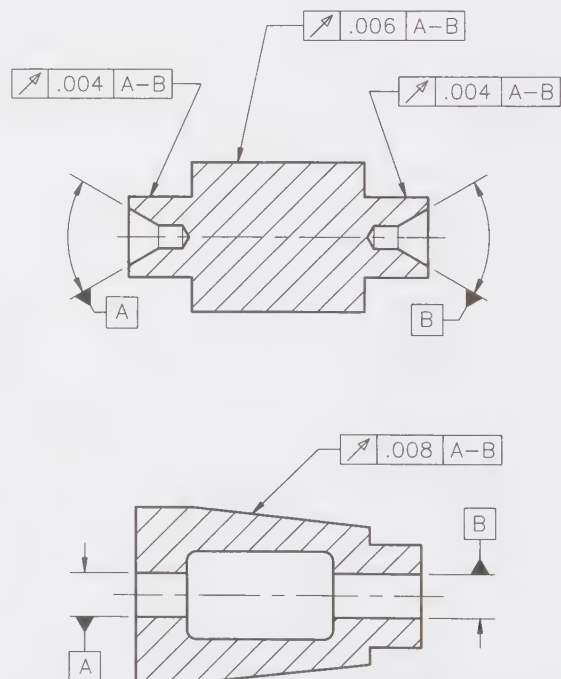
It would be incorrect to reference only datum A or only datum B for this part, because the function of the part does not use only one of the features. The proportions of the part would make it unlikely that proper location of the datum axis could be established from only one of the datum features. The given figure includes runout tolerances

applied to the two large diameters. The runout tolerance on each surface is to be measured while rotating the part on the axis established by datum features A and B.

Multiple datum features may be internal features such as the countersinks of centerdrilled holes or coaxial holes. See **Figure 10-9**. Centerdrilled holes are seldom functional features in a design and therefore are not always a good choice for datum features. However, centerdrilled holes are often used in production and are therefore considered a practical choice for datum features.

It is possible to use nonfunctional features such as centerdrilled holes as datum features. To do this with success, the accumulation of tolerances between the nonfunctional and the functional features must be considered. The given part has runout tolerances of .004" on the two bearing surfaces at the end of the shaft, and a runout tolerance of .006" on the large diameter. These tolerances are all referenced to the same datum axis established by the two centerdrilled holes. The worst-case runout variation between the two bearing diameters and the large diameter is equal to the sum of the tolerance on the features, in this case .010".

The datum axis for this part is established in manufacturing by using two machine centers. A machine center is placed in each centerdrilled hole, and the part is rotated.



Goodheart-Willcox Publisher

Figure 10-9. Multiple datum feature references may be used to establish a single axis of rotation through two datum features.

Coaxial holes in a part may be used to establish a datum axis. The second illustration in **Figure 10-9** shows a runout tolerance referenced to a datum axis that is established from two coaxial holes. Each of the shown coaxial holes has a datum feature symbol attached. The runout tolerance specification references the two datum features.

An expanding mandrel can be used to locate the datum axis. A fixed-diameter shaft must not be used (unless it fits tightly), because datum feature references for runout tolerances are always RMB. Dial indicator readings are taken while the part is rotated on the datum axis established by the coaxial holes.

Datum targets are used whenever specific locations on a feature are used to establish a datum. There are many plausible reasons for using datum targets. One reason is that a functional datum feature may be large enough to make contact with the whole feature impractical. Another reason is that a feature may be irregular, and therefore require that specific locations be selected.

Whatever the reason, datum targets as defined in a previous chapter may be used with runout tolerances. See **Figure 10-10**. Datum target lines at the two ends of the part are used to define datum axis A-B on the given part. The runout tolerance is referenced to multiple datum features A-B, and those two features establish datum axis A-B.

Because the target lines are equally spaced on each end, two machine chucks of adequate precision may be used to establish the datum axis. Three-jaw chucks must be used because three line contact is required.

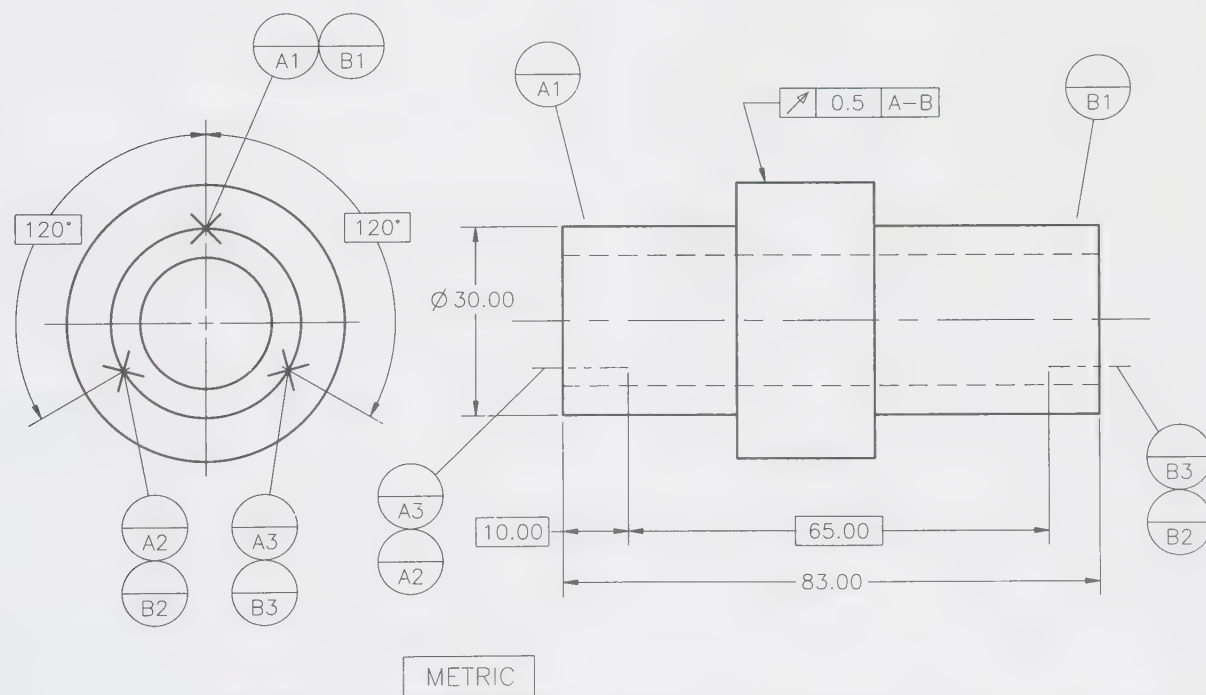
Limited Tolerance Zone Application

Tolerances should only be applied to establish the amount of control required for the parts to properly function. Specifying only the needed level of control prevents the cost increases associated with over-specification of tolerances.

Runout tolerances control the entire surface to which they are applied unless indicated otherwise. When only a portion of a surface requires a runout tolerance, the application of the tolerance can be limited. See **Figure 10-11**.

A thick chain line is used to show the area of application for a runout tolerance. The chain line is drawn along the outside of a view that shows the segment of the surface to be controlled. The chain line location and length are dimensioned with basic dimensions. If one end of the chain line is at the edge of the tolerated surface, no location dimension is needed.

The chain line only indicates a limit of application in one direction. The runout tolerance still applies all the way around the circular elements that lie within the specified length.



Goodheart-Willcox Publisher

Figure 10-10. Runout tolerances may be referenced to a datum axis that is established from datum targets.

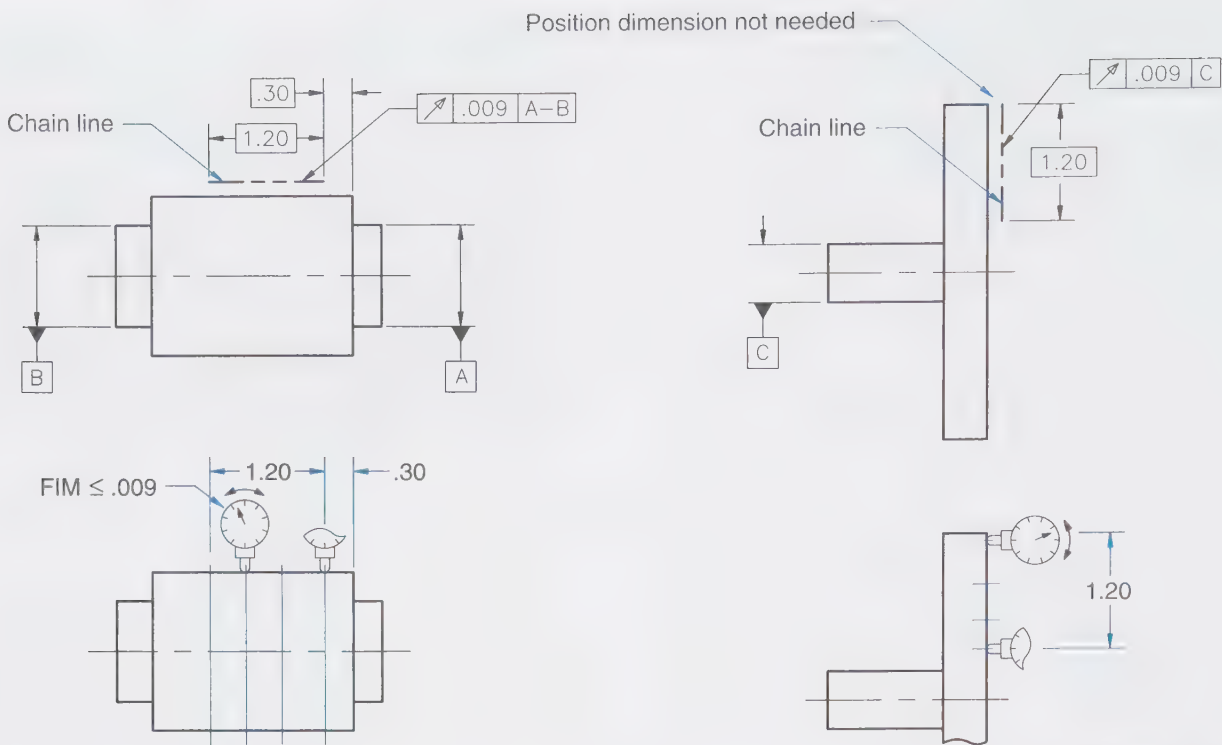


Figure 10-11. A limited zone of application for a runout tolerance is indicated by a heavy chain line.

Runout Referenced to Two Datums

One datum feature reference is adequate for many applications, but some applications require a second datum to stabilize the part. Runout tolerances having a primary and secondary datum feature reference must include one datum axis and one datum plane. Only the design function determines whether the axis or the plane is referenced as primary. It is permissible for either the axis or the plane to be primary.

Selection of the correct primary datum is important. The primary datum will establish the orientation of the part and will have a significant impact on the produced part.

Primary Axis, Secondary Plane

An axis is usually chosen as the primary datum for runout tolerances when the dimensioned part is mounted on the diameter of a shaft or inserted in a hole. The cylindrical datum feature should be primary in these situations because the diameter of the feature serves to orient and locate the part.

See **Figure 10-12**. The primary datum feature reference on the given part establishes an axis. The secondary datum feature reference establishes a datum plane that is perpendicular to the axis.

Primary datum axis A, for the given part, is established from datum feature A. Datum feature A

is a cylinder. Datum feature A has a length-to-diameter ratio that may not adequately stabilize the part when clamped in a chuck or collet. Secondary datum feature B is used to stabilize the part on datum axis A. The datum axis is both located and oriented by primary datum A. Datum feature B is only used to maintain the axis.

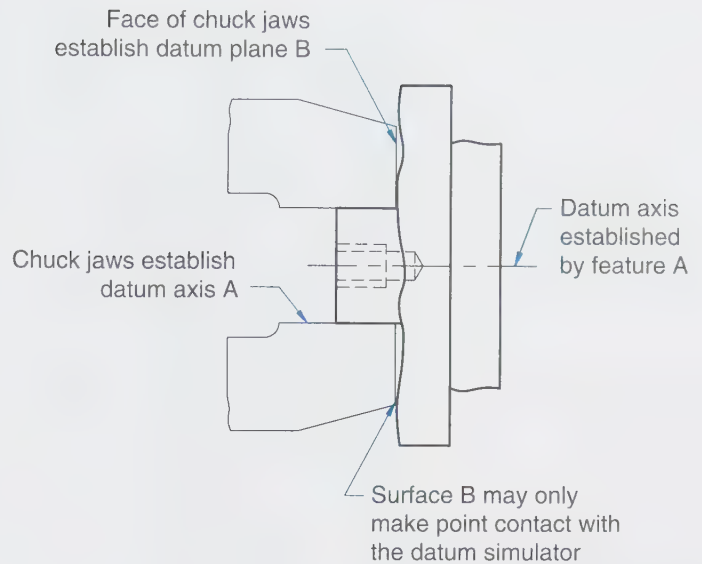
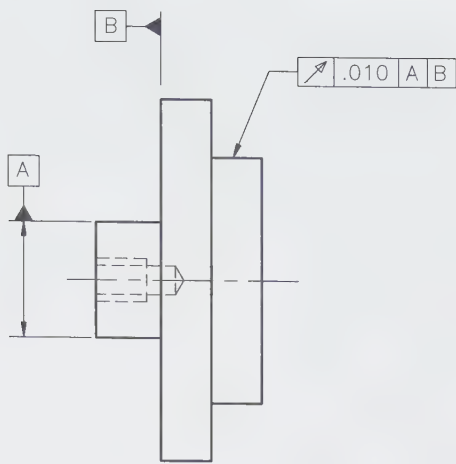
Primary Plane, Secondary Axis

Some types of rotating parts are mounted on a face surface. A flywheel mounted on a crankshaft is one example. Because the part mounts on a face surface, the face surface establishes the orientation for the axis of rotation. Another feature is required to locate the axis of rotation.

See **Figure 10-13**. A gear blank is shown in the given figure. The face of the completed gear will bolt against the face of another gear. A pilot on the shown gear blank fits into a hole in the other gear.

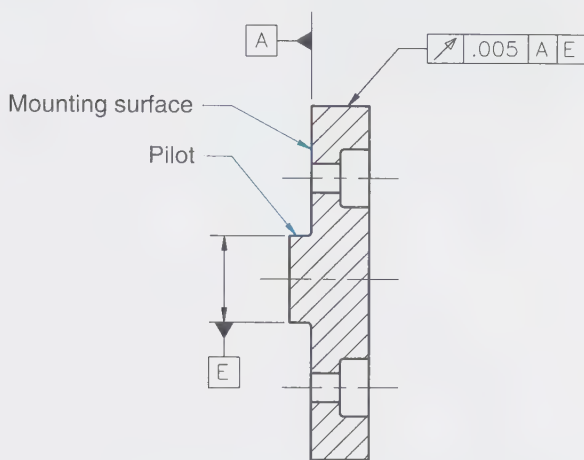
The gear face is identified as datum feature A, and the pilot is identified as datum feature E. The runout tolerance applied to the gear blank outside diameter references datum plane A (primary) and datum axis E (secondary).

The face of the gear establishes orientation relative to the axis of rotation. The pilot serves to locate the axis. Proper simulation of the functional application for this part requires the flat surface be used as the primary datum.



Goodheart-Willcox Publisher

Figure 10-12. A secondary datum may be used to stabilize a part on the primary datum axis.



Goodheart-Willcox Publisher

Figure 10-13. The mounting surface for a part sometimes requires that the primary datum be a plane. The datum axis is located by a secondary datum feature reference when runout is specified relative to a primary datum plane.

Total Runout

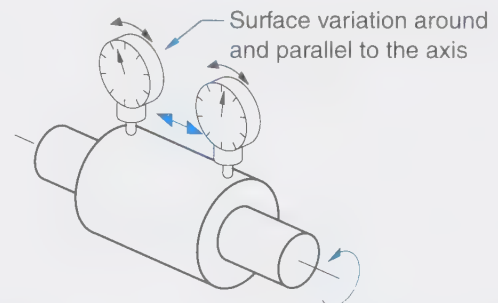
Total runout is the variation across the entire surface of a cylindrical feature or a perpendicular face surface. Application on other surface shapes is not recommended. Use on other shapes is likely to create confusion regarding requirements, because there is no standardized meaning for total runout on anything other than cylinders and flat surfaces. Profile tolerances are recommended for other shapes.

The allowable form and location relative to an axis of the cylindrical or flat surface elements are included in the total runout tolerance. So, the roundness, straightness, location, and orientation

of a cylinder are controlled relative to the specified datums. When total runout is applied on a surface that is perpendicular to the datum axis, the flatness and orientation of the surface elements are controlled. So, the total runout will prevent wobble of the face surface on a rotating part. The total runout tolerance value is not limited by the size limits, and size limits are not affected by a total runout tolerance.

All elements on the surface are evaluated with one measurement value. The surface variations are measured relative to an axis of rotation. See [Figure 10-14](#). The given part is a simple step shaft that rotates on surfaces at the two ends of the part.

The total runout of the large cylinder is measured by moving a dial indicator parallel to the axis of rotation while rotating the part on the axis of rotation. The full indicator movement caused by the surface variations along the feature is the total runout variation.



Goodheart-Willcox Publisher

Figure 10-14. The combined effect of all surface variations on a feature, relative to an axis of rotation, is the total runout.

Feature Control Frame

A total runout tolerance specification has a required format. See [Figure 10-15](#). The total runout tolerance symbol is shown first; it is a double arrow. The arrowheads may be filled or unfilled. The tolerance value follows the symbol. A diameter symbol is never used because runout tolerances indicate surface variations. A diameter symbol would incorrectly indicate a cylindrical tolerance zone.

As explained for circular runout, total runout tolerances are always applied RFS. Datum feature references are made in the same manner as for circular runout and always apply RMB.

Total Runout Applications

Cylindrical surfaces and flat surfaces perpendicular to the datum axis may be controlled with total runout tolerances. Total runout is not typically applied to a curved or tapered surface. Total runout controls the form, orientation, and location (coaxiality) of the controlled surface relative to a referenced datum or datums.

Total Runout Applied to a Cylinder

The feature control frame for total runout may be applied to cylindrical features in any of several ways, all of which indicate the same requirement on the indicated feature. See [Figure 10-16](#). The feature control frame may be attached to the surface using a leader. Another option is to attach the feature control frame to an extension line from the surface. The feature control frame may also be associated with the size dimension through placement near the dimension value or attachment to the dimension line, but this practice is no longer recommended because of industry requests that this method be removed from the ASME standards.

Two parts are shown in the given figure. They show that total runout tolerances, like circular

runout tolerances, may be controlled relative to internal or external datum features. One of the parts is a shaft that has a bearing diameter at each end. These are identified as datum features A and B. The total runout tolerance on the shaft is referenced to multiple datum features A-B.

Total runout on a part produced to the given drawing may be checked through the following procedure. Datum features A and B must be used together to establish an axis of rotation. This may be accomplished by clamping the two features in collets or chucks. Continuous dial indicator readings must be taken along the length of the part as it is rotated. The feature is acceptable provided the full indicator movement is equal to or less than .005". The photograph shows an alternate means of establishing the datum axis using V-blocks. This method is adequate for explaining the concept and may be used when the datum axis simulation is not critical. The datum axis established by V-blocks may have some inaccuracy if the datum features are not true cylinders.

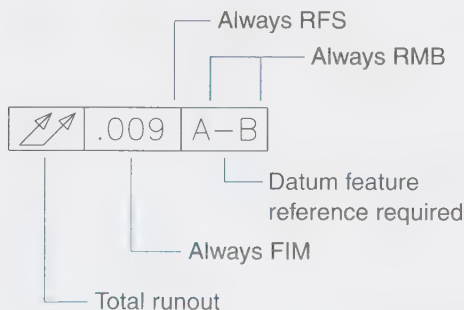
In the given example of the internal datum feature, the datum axis is established by an expanding mandrel. The mandrel expands to find the surface of the datum feature regardless of the feature size.

Total runout may also be applied to internal features. The interpretation is the same as when applied to external features. Total runout applied to an internal cylinder requires that the full indicator movement across the entire surface be equal to or less than the specified tolerance value.

Total Runout Applied to Face Surfaces

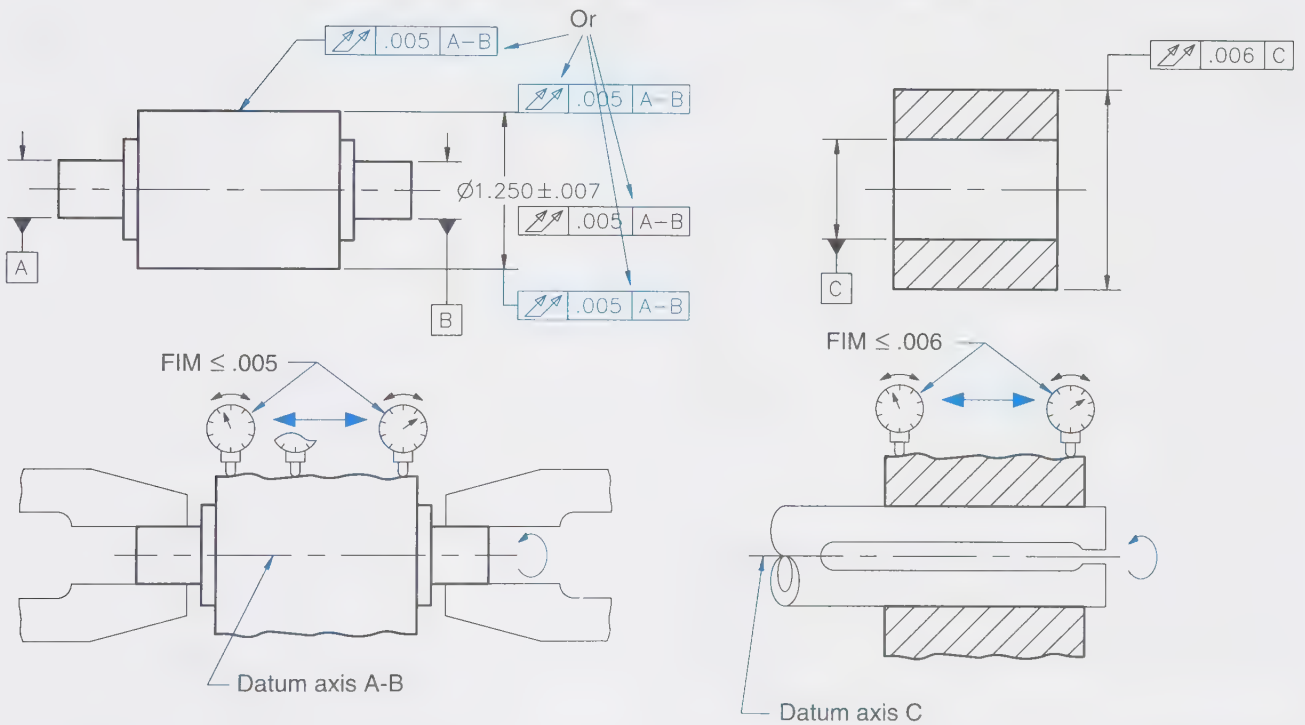
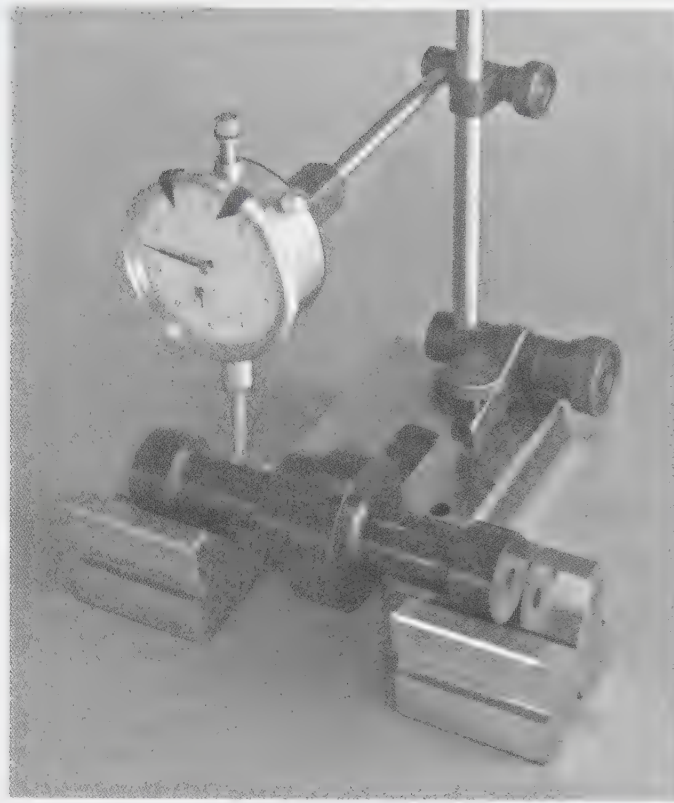
Total runout tolerances may be applied on extension lines from face surfaces or connected to them with a leader. Both application methods result in the same control for the surface. The tolerance must not be applied to a size dimension when the intent is to specify a total runout tolerance on a flat surface.

See [Figure 10-17](#). The given figure shows a .009" total runout tolerance applied to a flat face surface. This requires that the surface be flat and properly oriented within the .009" tolerance when the part is rotated on the datum axis. This is checked by positioning a dial indicator perpendicular to the surface and moving the dial indicator across the surface as the part is rotated. The dial indicator is moved across the face surface in a direction perpendicular to the axis of rotation. If the indicator readings show .009" or less variation, then the total runout tolerance specification has been met.



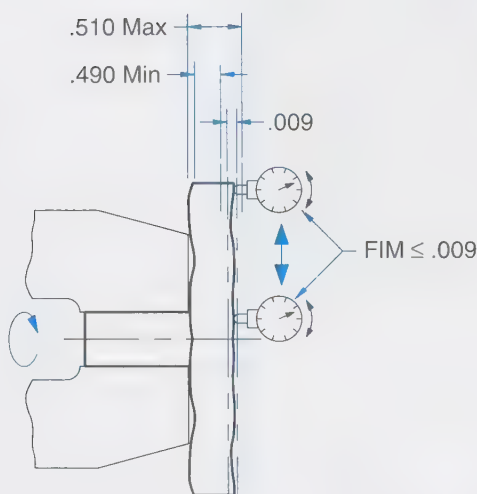
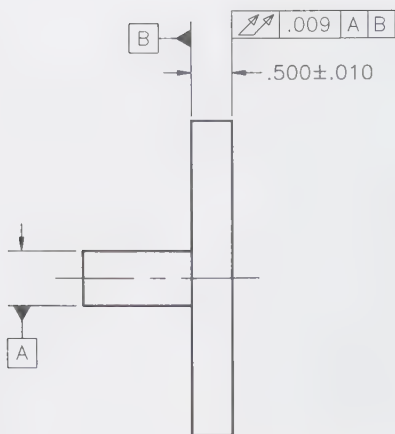
Goodheart-Willcox Publisher

Figure 10-15. Present practice requires a double arrow be used to indicate total runout.



Goodheart-Willcox Publisher

Figure 10-16. A feature control frame for total runout may be applied in any of several methods and have the same meaning.



Goodheart-Willcox Publisher

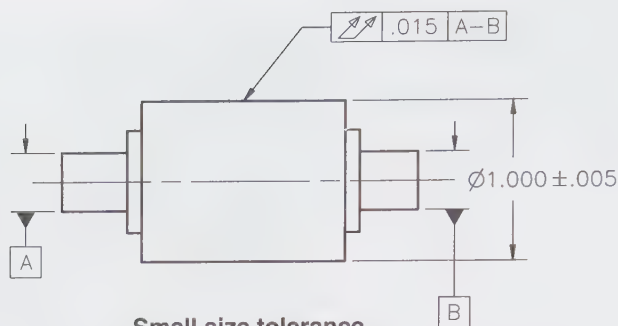
Figure 10-17. Total runout applied to a face surface should be applied on an extension line from the surface or attached to the surface using a leader line.

The shown total runout tolerance only applies to one surface. Datum feature B is not affected by the runout tolerance.

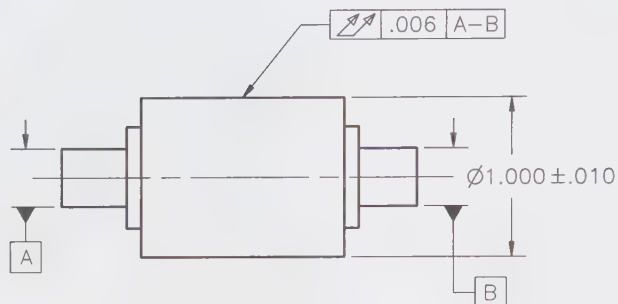
Combined Effects of Size and Runout Tolerances

Runout tolerances applied to a feature may be larger than the size tolerance applied to that feature. See **Figure 10-18**. It is also acceptable to apply a runout tolerance that is smaller than the size tolerance. Neither the runout tolerance nor the size tolerance is constrained by the other.

Runout variation is possible even when a feature has perfect form. See **Figure 10-19**. The given figure has a dimension of $1.000'' \pm .005''$ on one feature. When this feature is at the MMC size of $1.005''$ diameter, it must have perfect form. The perfect form requirement at MMC does not control the location of the feature relative to other features on the part.



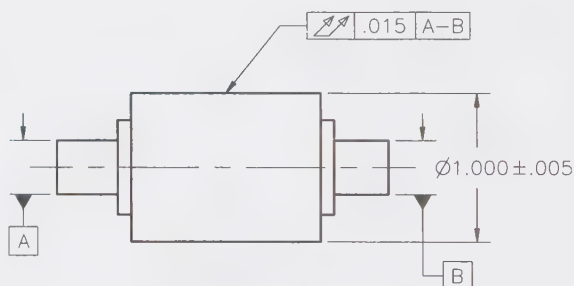
**Small size tolerance,
large runout tolerance**



**Large size tolerance,
small runout tolerance**

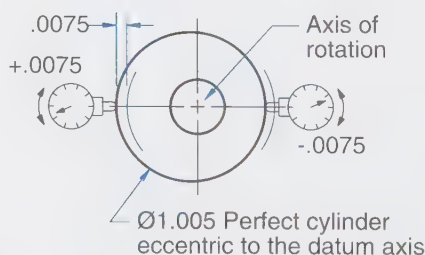
Goodheart-Willcox Publisher

Figure 10-18. A runout tolerance applied to a feature may be larger or smaller than the size tolerance on the feature.



**Small size tolerance,
large runout tolerance**

FIM ≤ .015



Goodheart-Willcox Publisher

Figure 10-19. A feature may be at MMC, have perfect form, and still include runout variation relative to the referenced datum axis.

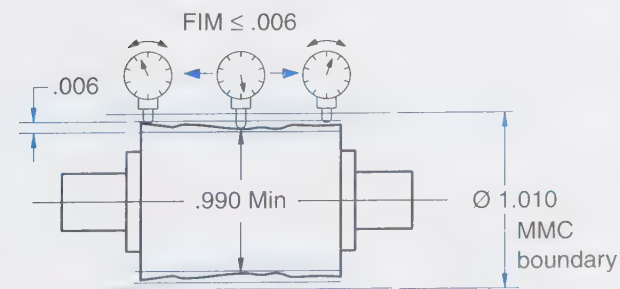
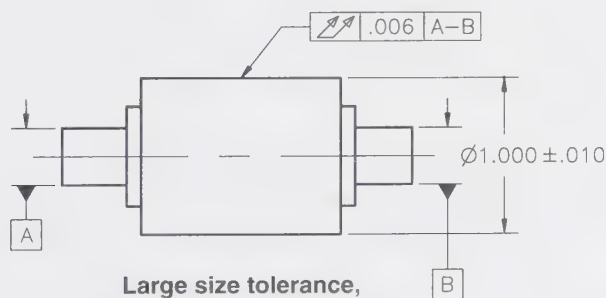
The given figure shows a permissible total runout tolerance of .015" relative to datum axis A-B. This allowable variation may be caused by a combination of size, orientation, and location variations. A feature produced at MMC has no size variations, but runout variations may still exist as a result of orientation and location variation.

One acceptable condition for a part produced to the given drawing is shown. It has a perfect form cylinder at the 1.005" MMC diameter. The cylinder is offset from the axis by a distance that results in a full indicator movement of .015". This variation of the surface relative to the datum axis is permitted.

Total runout variation is bounded by two concentric cylinders. These cylinders must contain the tolerated surface and may be any diameter provided the radial separation between the cylinders is no greater than the specified total runout tolerance.

Size limits are not controlled through the application of total runout tolerances. See **Figure 10-20**. The given figure has a size tolerance of .020". It also has a total runout tolerance of .006". This shows that the limits of size may be assigned a larger permissible tolerance than the total runout tolerance.

The 1.000" cylinder on the given part may be produced at any diameter between .990" and 1.010". The cylinder must not exceed the 1.010" MMC envelope. Whatever the produced diameter of the cylinder, it must not have surface variations more than .006" relative to the datum axis.



Goodheart-Willcox Publisher

Figure 10-20. Runout tolerances are not meant to control the limits of size; they do control the amount of permissible surface variation.

It is possible to produce a part that has at least one cross-sectional measurement that is at the .990" minimum diameter. A part produced with one .990" diameter cross section is not permitted to have any other cross section that is larger than 1.002" diameter. This is because the total runout tolerance only permits a .006" full indicator movement. The .006" full indicator movement permits a maximum of .012" variation on the diameter of a produced part. If a part is produced with a .994" minimum cross-sectional measurement, then its maximum permitted cross-sectional measurement is 1.006" diameter.

Chapter Summary

- ✓ Circular runout tolerances control each of the circular elements on an entire surface.
- ✓ Total runout tolerances simultaneously control all elements on a surface.
- ✓ Runout tolerances apply to an entire surface unless a limited application zone is shown.
- ✓ Runout tolerances specify allowable surface variations relative to an axis of rotation.
- ✓ Runout tolerances are always applied RFS and the datum feature references for runout tolerances are always RMB.
- ✓ The tolerance value for a runout tolerance is the maximum allowable full indicator movement.
- ✓ The amount of runout tolerance applied to a part is not directly affected by the size tolerances on the controlled features.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. _____ requires that only individual circular elements meet the specified tolerance value.
 - A. Total runout
 - B. Circular runout
 - C. Face runout
 - D. None of the above.
2. A runout tolerance always includes _____ datum feature reference(s).
 - A. one or more
 - B. one
 - C. two
 - D. multiple

3. Circular runout is measured at _____ location(s) on the controlled surface.
 - A. one
 - B. multiple
 - C. five
 - D. None of the above.
4. A cylinder may include acceptable variations that result in a tapered (conical) shape if _____ runout is applied. The amount of taper may exceed the specified runout value.
 - A. total
 - B. circular
 - C. Either A or B.
 - D. Neither A nor B.
5. One method of checking runout tolerance requires the part to be _____ with a dial indicator in contact with the controlled feature.
 - A. held stationary
 - B. moved parallel to its axis
 - C. rotated on its axis
 - D. None of the above.
6. Circular runout tolerances may be applied to any feature that has _____.
 - A. a cylindrical shape
 - B. a flat shape perpendicular to the axis
 - C. circular elements related to an axis of rotation
 - D. All of the above.
7. A _____ zone is created by drawing a chain line along a portion of the surface to which a runout tolerance is applied.
 - A. limited application tolerance
 - B. tolerance
 - C. projected tolerance
 - D. None of the above.
8. Circular runout tolerances _____ applied to a tapered shaft.
 - A. control taper when
 - B. cannot be
 - C. control roundness when
 - D. should reference three datums when
9. Runout applied to a face surface controls _____.
 - A. wobble
 - B. radial movement
 - C. eccentric cam motion
 - D. None of the above.

True/False

10. *True or False?* A dial indicator does *not* indicate the diameter of the object being measured.
11. *True or False?* The primary datum feature for a runout tolerance must never be a flat surface.
12. *True or False?* Each measurement for circular runout is considered separately.
13. *True or False?* The surface variations permitted by a runout tolerance do not affect the size limits on a controlled surface.
14. *True or False?* Runout tolerance may be applied to internal and external surfaces.
15. *True or False?* Datum targets may be used to establish a datum axis.
16. *True or False?* When a runout tolerance references two datums, the primary datum must establish an axis.
17. *True or False?* Runout variation may exist relative to a datum axis even when a cylinder has perfect form.

Fill in the Blank

18. The symbol for total runout has _____ arrowhead(s).
19. Runout variations are always measured relative to a datum _____.
20. One continuous measurement is taken to check _____ runout of a feature.
21. Runout tolerance may be shown in a feature control frame that is placed adjacent to the size dimension for the feature, on an extension line from the surface, or attached to the surface with a(n) _____.
22. A part that rotates on two bearings should have a runout tolerance that references a(n) _____ datum.
23. Total runout tolerances may be applied to flat surfaces only when the surface is _____ to the axis of rotation.
24. Total runout applied to a cylinder creates a tolerance zone bounded by two _____ cylinders.

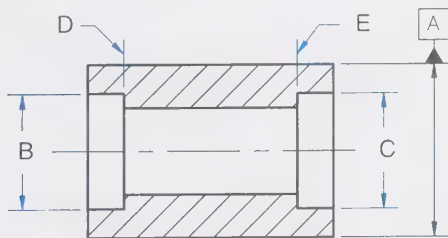
Short Answer

25. What is meant by the phrase *full indicator movement*?
26. Explain the effect of referencing multiple datum features as a primary datum.
27. Is it permissible to specify a circular runout tolerance on a conical feature? Explain your answer.
28. Why is the application of total runout on a spherical feature not typically done?
29. How can circular runout of a tapered shaft be checked?

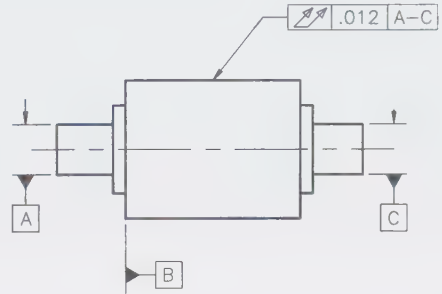
Application Problems

Each of the following problems requires that a sketch be made of the figure. All sketches should be neat and accurate. Apply all required dimensions in compliance with dimensioning and tolerancing requirements. Show any required calculations.

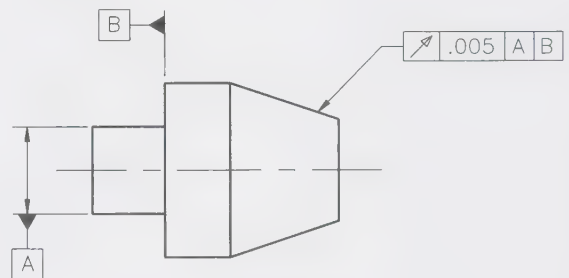
30. Sketch a feature control frame that defines an allowable total runout of .009" relative to a datum axis that runs through datum features A and B.
31. Sketch a feature control frame that defines an allowable circular runout of .005" relative to datum axis D.
32. Control diameters B and C with a total runout tolerance of .005" relative to datum A. Control face surfaces D and E with a circular runout tolerance of .006" relative to datum A.



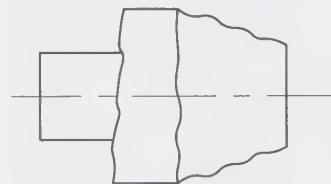
33. Sketch an inspection setup that includes a means of establishing the datum axis and measuring runout variation. Show some permitted surface variation on the feature that has the runout tolerance applied and also show the tolerance zone that must contain that variation.



34. Sketch the shown produced part in an inspection setup for measuring the specified runout tolerance. Include datum simulators and also show how measurements would be made.

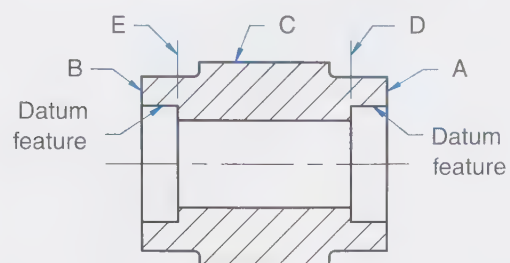


Drawing



Produced part

35. Apply a total runout tolerance of .006" to surfaces A, B, D, and E. Also apply a circular runout tolerance of .005" to surface C. Identify the two datum features and reference them as multiple datum features in the runout tolerance specifications.



Chapter 11

Profile

Objectives

Information in this chapter will enable you to:

- ▼ Define line and surface profile tolerances.
- ▼ Apply profile tolerances to define allowable variation within a limited zone on a feature or for all of a feature.
- ▼ Apply profile tolerances to extend all around the profile shown in a drawing view.
- ▼ Complete profile tolerance specifications to achieve any of the possible levels of control.
- ▼ Sketch the tolerance zone created by profile tolerance specifications.
- ▼ Specify coplanarity requirements using profile tolerances.
- ▼ Identify profile tolerances as the means for specifying allowable variation for conical surface form, orientation, and location.
- ▼ Draw a composite profile tolerance specification.

Technical Terms

bilateral profile tolerance
line profile
profile tolerance
surface profile
true profile
unequally disposed profile tolerance
unilateral profile tolerance

Introduction

Profile tolerances may be used for a wide variety of controls. These tolerances may be used to define allowable size, form, orientation, and location variation. The manner in which the profile tolerance is applied, if the toleranced feature is

dimensioned for location or size, and how datums are utilized all impact the level of control achieved through the use of profile tolerances.

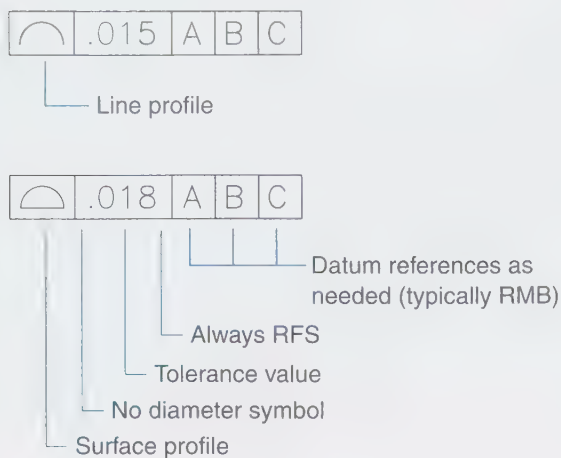
Because of the flexibility in the levels of control that may be achieved with profile tolerances, they are often used when no other tolerance control is appropriate. The flexibility that is provided by profile tolerances makes possible the specification of the exact needed control, but the flexibility in application also makes mastery of profile tolerance utilization somewhat challenging.

Subtle differences in how profile tolerances are applied to a drawing or in a model can make significant differences in the required tolerance requirements. Care must be used in the application of profile tolerances to ensure specification of the desired control.

Profile Specification

There are two distinctly different types of profile tolerances defined by ASME Y14.5-2009: profile of a line and profile of a surface. For simplification purposes, the terms line profile and surface profile are used here. *Line profile* is specified using a single curved line as the tolerance symbol. See [Figure 11-1](#). *Surface profile* is specified using a curved line with a horizontal line drawn across the bottom to close the symbol.

Feature control frames for the two profile tolerance types have the same format, except for the tolerance symbol. The profile tolerance value never includes a diameter symbol, and the tolerance value is always applied RFS. Depending on the situation, datum feature references may not be needed in a profile tolerance. When they are not needed, no datum references are shown. It is often the case that one or more datum feature references

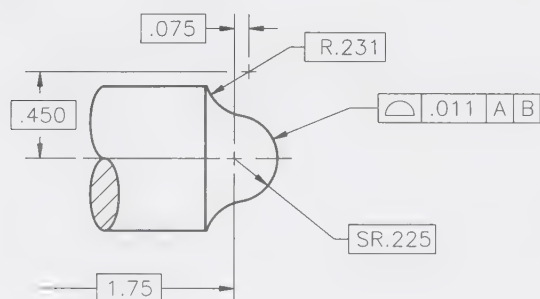


Goodheart-Willcox Publisher

Figure 11-1. Tolerance values for line profile and surface profile default to apply at RFS.

are needed, and they are shown in the feature control frame following the tolerance value. When the datum feature references are features of size, they are typically referenced RMB. The datum feature references may be at MMB or LMB. See Chapter 6 to review the effects of referencing datum features RMB, MMB, or LMB.

Several variables are involved in determining the correct application of a profile tolerance to a feature. If the profile tolerance is applied to a feature that is anything other than flat, then the form of the feature must be defined with basic dimensions or the CAD data noted as being basic. See **Figure 11-2**. The curved end on the shown forming punch is controlled by a profile tolerance. The curve must therefore be defined with basic dimensions. The two radii that form the punch are both shown basic. The relative location of the two radii centers is also given with a basic dimension. The four basic dimensions completely define the form of the curved surface.



Goodheart-Willcox Publisher

Figure 11-2. Profile tolerances control the shape (form) of the feature to which they are attached. Shapes controlled by profile tolerances are defined by basic dimensions.

A surface profile tolerance is attached to the curved surface with a leader. The tolerance specification includes two datum feature references; therefore, the tolerance controls the location and orientation relative to the datums as well as the form and size of the toleranced features. The extent of the requirements is established by a tolerance boundary that is defined relative to a theoretically perfect shape, typically called the *true profile*, created by the basic dimensions on the drawing. The term *nominal* is commonly used in industry for the true profile. However, the use of the term nominal is not precisely correct.

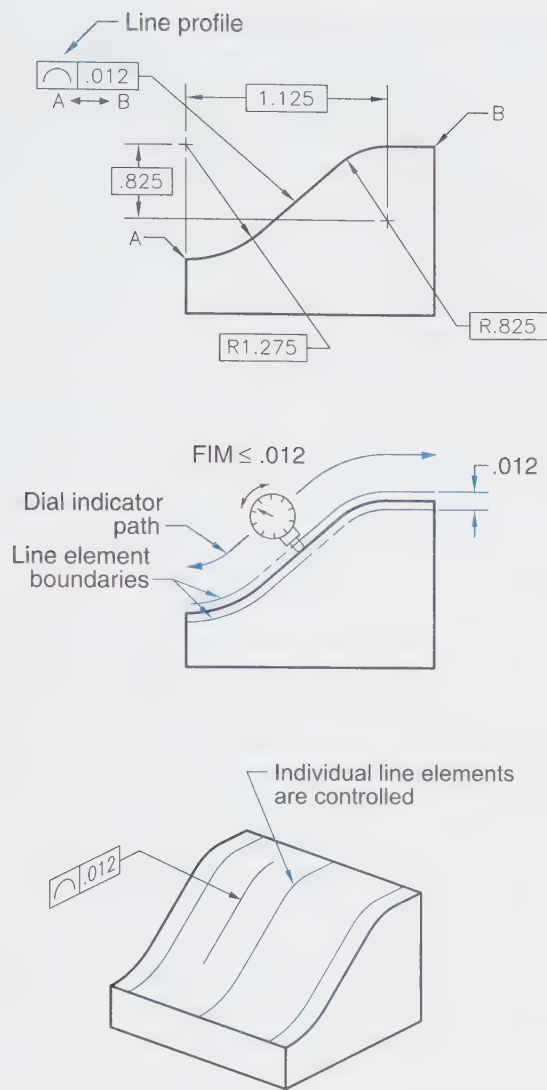
A general note may be used to indicate that a surface profile applies to all features that do not have tolerances directly applied. The noted surface profile applies to all features that are not toleranced by other means. Examples include holes that have size and position tolerances applied, as well as surfaces that have a surface profile attached by leader. It is also possible to refine the profile tolerance through the application of an orientation or form tolerance that is smaller than the noted profile tolerance. A note such as the following is commonly used on drawings and with CAD models.

A SURFACE PROFILE OF .038 RELATIVE TO DATUM A PRIMARY, B SECONDARY, AND C TERTIARY IS APPLICABLE TO ALL FEATURES UNLESS OTHERWISE SPECIFIED.

Line Profile

Line profile is used to control single line elements on a feature. The level of control invoked on the feature depends on what is shown in the feature control frame.

The simplest level of control that may be specified with a line profile tolerance is to control only form of line elements. See **Figure 11-3**. It should be noted that this figure is only complete to the extent necessary to explain a line profile tolerance that has no datum feature references. The shown tolerance specification would not by itself establish adequate requirements to fully define the allowable line element location and orientation tolerances. Additional tolerances would be needed to establish those additional requirements. The upper portion of the figure shows a line profile tolerance of .012" applied to the curved surface in an orthographic view. Each line element that is parallel to the shown profile must have a form that lies within a .012" wide tolerance zone. Because no datum features are referenced in the tolerance specification, the .012" wide boundary has no required orientation or location relative to any



Goodheart-Willcox Publisher

Figure 11-3. Line profile tolerances control individual line elements that are parallel to the view in which the tolerance is specified.

datum reference frame. In fact, no datum reference frame is invoked. The .012" wide profile tolerance zone controls only the form of individual line elements on the surface. The location and orientation of the surface must come from other tolerances, perhaps a surface profile tolerance that includes datum feature references as explained later in this chapter. It is also possible to add datum feature references to this feature control frame to further establish location and orientation requirements for the tolerated surface.

Each surface element parallel to the profile on which the tolerance is applied has a .012" wide zone. Each element is independent of all others. There are no form requirements established in any direction other than parallel to the profile to which the tolerance is applied.

Application of a line profile tolerance in a CAD model requires that a directed line element be shown. The solid model in the lower portion of the figure shows a directed line element. The directed line element is created by the intersection of a plane and the surface. No length requirement is standardized. Extending the line the full length of the surface is not recommended because that could appear to indicate an edge. The feature control frame is placed on the same plane containing the directed line element, and a leader extends from the feature control frame to the directed line element. The leader terminates with an arrow-head. The figure shows additional lines on the surface to indicate all parallel line elements on the surface are tolerated (these additional lines are not included in the CAD model).

Surface Profile

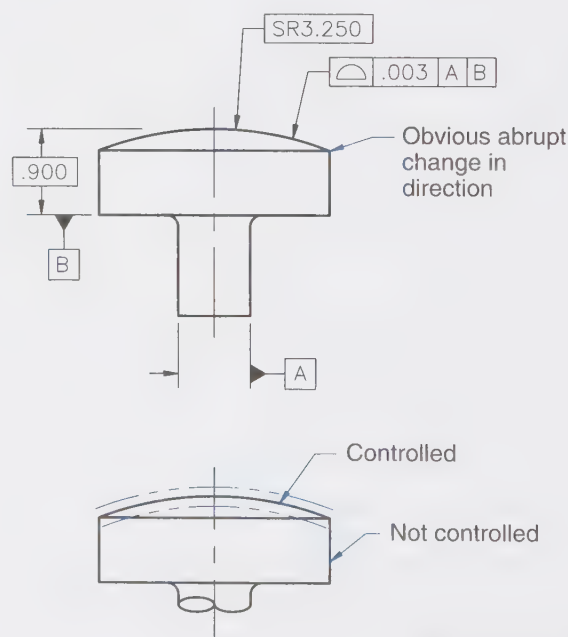
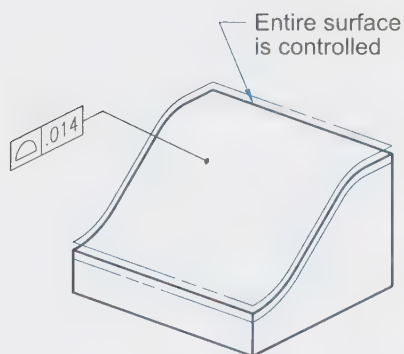
Surface profile is used to simultaneously control all elements on a feature. Whether form, form and orientation, or form, orientation, and location are controlled depends on what is shown in the feature control frame. When applied to one surface on a feature of size (such as one side of a slot), the size tolerance also affects the surface requirements.

The simplest level of control that may be specified with a surface profile tolerance is to control only form. See **Figure 11-4**. It should be noted that this figure is only complete to the extent necessary to explain a surface profile tolerance that does not include datum feature references. The shown tolerance specification would not establish adequate requirements to fully define the allowable surface location and orientation tolerances.

The upper portion of the figure shows a surface profile tolerance of .014" applied to the curved surface. All points on the surface defined by the shown profile must lie within a .014" wide boundary. The boundary is centered (equal bilateral) on a perfect form profile (true profile) defined by the basic dimensions that define the curved surface.

Because no datum features are referenced in the tolerance specification, the .014" wide boundary has no required orientation or location relative to any datum reference frame. In fact, no datum reference frame is invoked.

The .014" wide profile tolerance zone controls only the form of the surface. The location and orientation of the surface may be defined in other tolerance specifications, perhaps another and larger surface profile tolerance that includes datum feature references as explained later in this chapter. It is also possible to add datum feature references to this feature control frame to further establish



The shown surface profile tolerance only applies to the spherical surface.

Profile Application Between Noted Limits

Abrupt changes in direction do not always exist at the desired limits of application for a profile tolerance, and sometimes people will have different interpretations of what an abrupt change in direction is. To overcome these issues, limits of application can be shown. Design function may at times require a profile tolerance that extends beyond abrupt changes in direction. For a profile tolerance to extend past the abrupt changes in direction, the limits of application must be shown.

Limits of application may be located anywhere on a part. See **Figure 11-6**. Limits of application are noted on an orthographic view of a drawing. In addition to labeling limits of application on the controlled surface, the limits are referenced below the profile tolerance specification. The “between” symbol is used to indicate the tolerance extents between the labeled limits of application. If the exact point location is significant, it may need to be located with dimensions.

Three profile tolerance feature control frames with limits of application are shown in the given figure. The surface profile tolerance of .010 is limited to the surface between points B and C. The surface profile tolerance of .020 extends past any abrupt changes in direction between points A and B and between points C and D. The profile tolerance extends between the referenced points in a direction that includes the surface where the tolerance leader makes contact.

The limits of application are identified by labeling points A, B, C, and D. Point C in the given figure is the point of tangency between a curved line and a straight line segment on the surface.

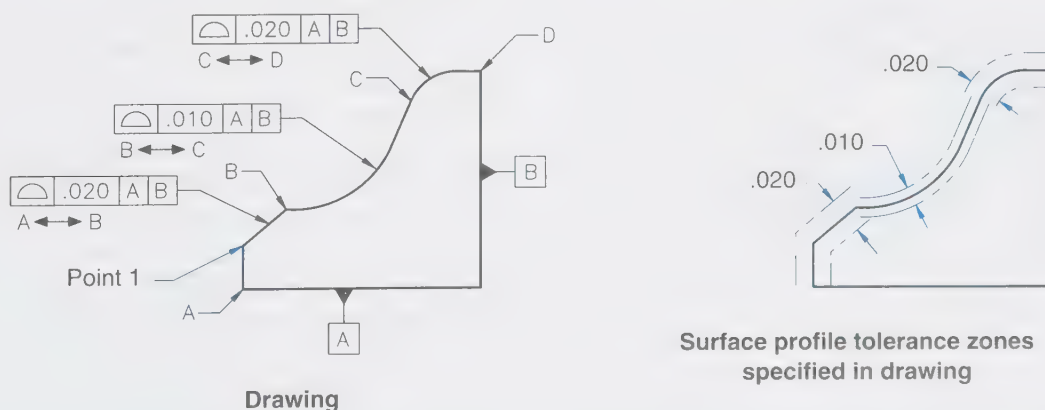
Point 1 is the intersection of two straight segments that are at slight angles to one another. Point C is definitely not an abrupt change in direction. It must be identified and referenced if the tolerance zone is to end at this point. The location of point C in the given figure may come into question, and a location dimension would clarify its location.

Point 1 may be considered an abrupt change by some people. Others would say that it is not an abrupt change. To prevent different interpretations of the tolerance requirements, points of application should be defined when there is no obvious abrupt change in direction.

Application to a limited area on a CAD model is standardized differently than for orthographic views. The feature or features that are included in the extent of application are associated with the profile tolerance specification. When the feature control frame in the model is selected (queried), the surface or surfaces are expected to highlight in accordance with ASME Y14.41. Although this is the standard method, not all CAD systems may support this, or the association between the feature control frame and features may be lost when the model is translated for use in another system. Should loss of association be a concern, labeling the extent of application in a manner similar to the method used in orthographic views may be a simple solution, although that method is not standardized for CAD models.

All Around Application

There are times when the profile tolerance needs to extend completely around the tolerated feature. These requirements may be met through the use of an all around symbol. In an orthographic view, the all around symbol indicates that the profile tolerance is to extend all the way around the profile of the feature to which the tolerance is applied.



Goodheart-Willcox Publisher

Figure 11-6. Limits of application may be labeled and referenced under the feature control frame.

The all around symbol is similar to the all around symbol used in welding specifications. See [Figure 11-7](#). It is a circle placed at the corner in the tolerance specification leader.

The shown profile tolerance has a 3X notation. The 3X notation may be located at the front of the feature control frame, but intent may also be made clear when it is placed above or below the feature control frame. The 3X in the given figure indicates that the tolerance applies to all three slots. For simplicity, the figure only shows the effect of the tolerance specification on one slot.

In a CAD model, the profile feature control frame is associated with the three features so that selecting the feature control frame results in all three slots highlighting. If the slots are made of multiple features, all the features are associated with the feature control frame so that all of them highlight. If loss of associativity is a concern, then the slots can be labeled, and a listing of the slot labels placed under the feature control frame.

The given drawing is a thin sheet metal plate with three slots cut in it. This part serves as a heat sink on a printed wiring board. The slots must

provide clearance for electronic component leads (wires). Size, form, orientation, and location of all slot surfaces must be properly tolerated to prevent the component leads from shorting against the metal heat sink.

A profile tolerance is applied to the slots to provide the desired level of control. Because one of the slots includes abrupt changes in direction, an all around symbol is placed on the profile tolerance specification. This symbol indicates that the tolerance zone extends all around the feature. Omission of the all around symbol would result in a tolerance zone that only extends between two of the abrupt changes in direction.

The all around symbol when used in an orthographic view only indicates a requirement for the tolerance to apply all around the feature profile as it is seen in the view to which the tolerance is applied. It does not indicate a requirement for the tolerance to apply over all surfaces of the part. In a CAD model, the intended direction of application for the all around symbol can be unclear unless the tolerated features are associated with the feature control frame and highlight when the feature control frame is selected.

All Over Application

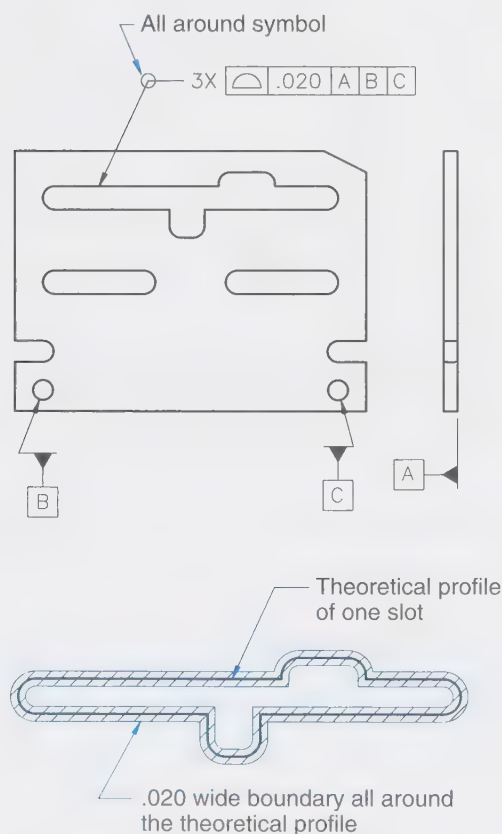
Design requirements may require a profile tolerance that applies to all features on a part. When the same requirements are to be applied on all surfaces, the profile tolerance may be applied with the all over symbol. See [Figure 11-8](#). The figure only shows the outer boundary of the profile tolerance for simplification purposes.

The all over symbol means that every feature surface gets the same tolerance. It is possible to apply additional tolerances to individual features to define additional requirements for those features. As an example, a profile tolerance of .060" may be applied all over a part, and a size tolerance and position tolerance may also be applied to a hole. Assuming the size and position tolerance are smaller than the .060" profile tolerance, the hole would be required to meet the smaller tolerance.

Profile Tolerance Zone Boundaries

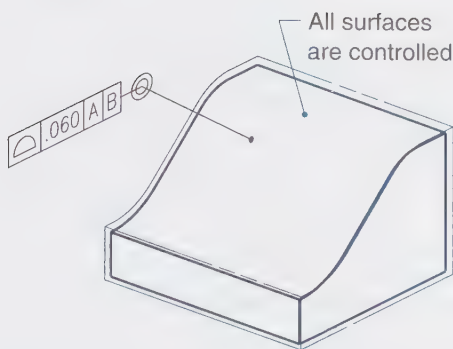
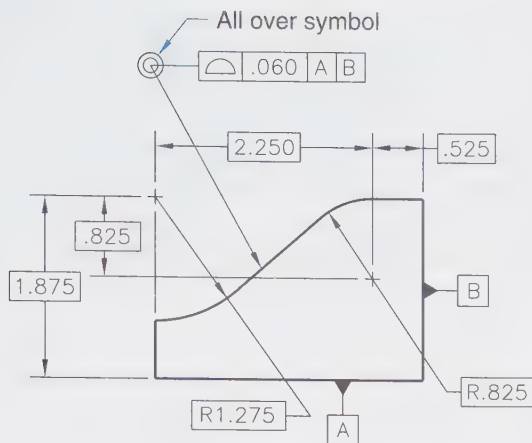
Profile tolerances default to create a tolerance zone that is centered on the true profile defined by basic dimensions unless otherwise specified. This has the effect of creating what is known as an equally disposed bilateral profile tolerance zone.

Profile tolerances may be specified in a manner that creates a unilateral profile tolerance zone or an unequally disposed profile tolerance zone. Unilateral profile tolerance zones are offset in only



Goodheart-Willcox Publisher

Figure 11-7. An all around symbol may be used to indicate that profile tolerances extend beyond abrupt changes in direction.



Goodheart-Willcox Publisher

Figure 11-8. An all over symbol may be used to specify a profile tolerance that applies to all surfaces on a part.

one direction relative to the true profile. Unilateral zones may be defined in a way that results in a plus tolerance (adds material) or a minus tolerance (reduces material). Unilateral zones are extreme conditions within a wider category of tolerance distributions known as unequally disposed profile tolerance zones.

Unequally disposed profile tolerance zones are specified in the profile feature control frame by using a symbol showing the letter U inside a circle. The method for specifying unilateral and unequally disposed tolerance zones was changed in 2009 with the 1994 method allowed as an alternate practice. Prior to 1994, the application of unequally disposed profile tolerances was not illustrated in the ASME standards. These methods will be further explained in the following paragraphs.

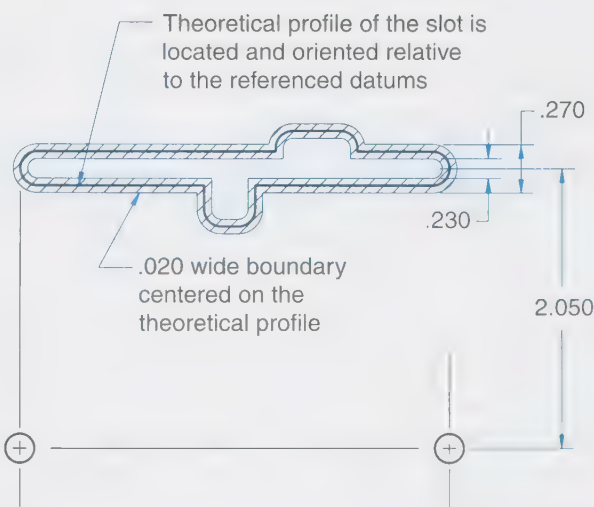
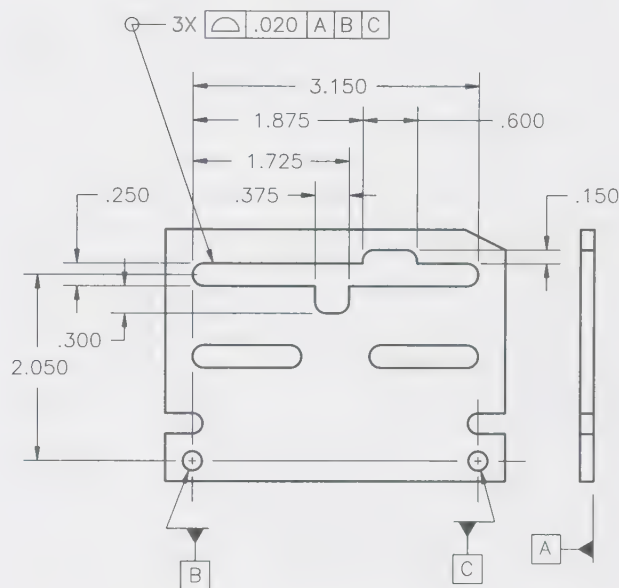
The application of the unequally disposed profile symbol is clearly defined and easy to apply in drawings and in CAD models. The previously defined practice of using a dimensioned phantom line to indicate the direction and magnitude of profile tolerance is considered an acceptable alternate practice but is not recommended.

Bilateral Profile Tolerance

An equally disposed profile tolerance that is centered on the basic profile is an equally disposed **bilateral profile tolerance**. The effect is that half the specified profile tolerance applies outside and half applies inside the true profile. Bilateral control is established when a profile tolerance is applied and no special indications are placed on the tolerance specification. See **Figure 11-9**. Size and location dimensions on the given slot are all known to be basic because a note in the figure

NOTES:

1. ALL UNTOLERANCED DIMENSIONS ARE BASIC.
2. ALL UNDIMENSIONED RADII ARE R.125



Goodheart-Willcox Publisher

Figure 11-9. Profile tolerances are assumed to be equally disposed bilateral unless indicated otherwise.

specifies this requirement. The basic dimensions establish a theoretically perfect true profile of the slot profile and a basic location. No tolerances are assumed applicable to the basic dimensions. The tolerances on the features must come from feature control frames or from the general notes. In the given example, the general notes do not specify any applicable tolerances.

The profile tolerance of .020" is referenced to datum features A primary, B secondary, and C tertiary. Because nothing indicates otherwise, a .020" wide tolerance zone is centered on the true profile and basic location defined for the slot. The slot surface may fall anywhere within the tolerance zone.

Size, form, orientation, and location of the slot are controlled by the profile tolerance. Size (the width of the slot) may vary by as much as $\pm .020$ from the basic dimensions shown in the drawing. Because the bilateral profile tolerance of .020 applies to each side of the slot, each side can vary by $\pm .010$. The distance between two sides can therefore vary by $\pm .020$. This allows the width to vary from .230" to .270" as shown in the interpretation segment of the figure. The location of the slot center is controlled by the profile tolerance. Based on the allowable tolerance boundaries shown in the figure, the slot center may vary by up to $\pm .010$ relative to the referenced datums, with the magnitude of location variation depending on the amount of the profile tolerance consumed by size and form variations.

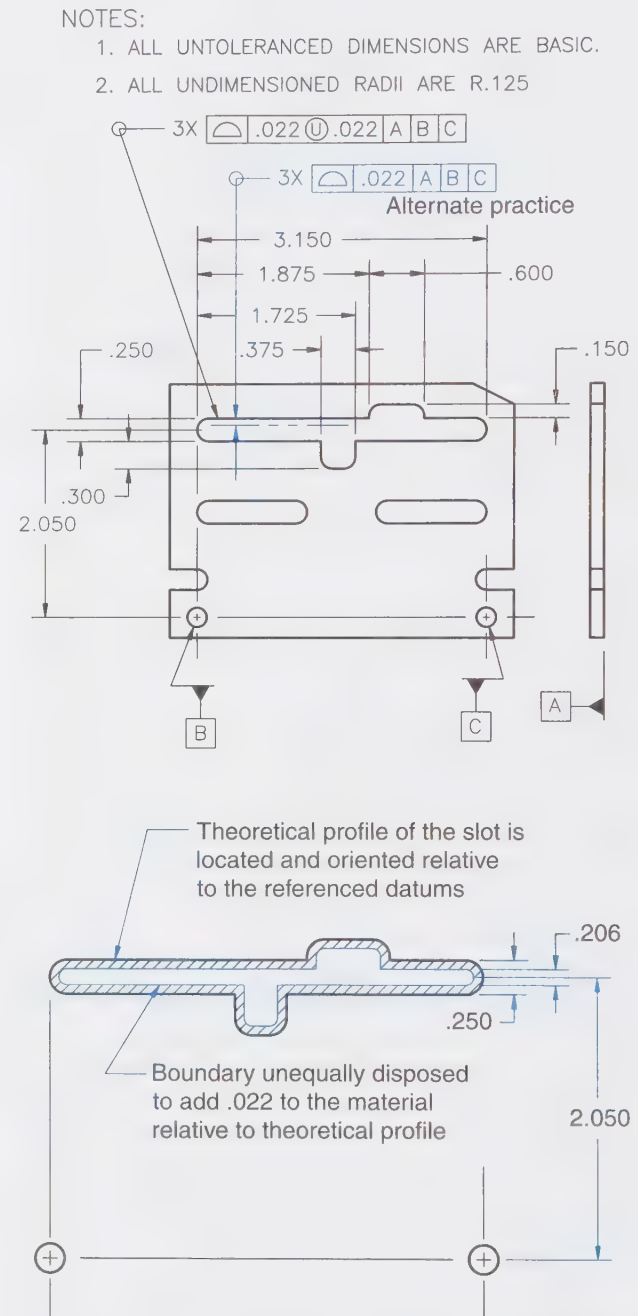
Although it may be commonly thought that the true profile is the expected fabrication target, no target is established by the profile tolerance. The true profile on which the tolerance zone is centered does not establish a required manufacturing target. If there is a needed manufacturing target for assembly or other design reasons (statistical tolerance calculations could be one reason), that requirement must be noted or required through a referenced process document.

Unilateral Profile Tolerance

An *unequally disposed profile tolerance* that is entirely on one side of the basic profile is a *unilateral profile tolerance*. It is defined with a feature control frame that includes the unequally disposed profile symbol (the letter U inside a circle). This symbol follows the tolerance value. Immediately following the unequally disposed profile symbol is a value that indicates the amount of the tolerance that is "outside the material" or would "add material."

Unilateral tolerances may be applied to create the effect of a tolerance that allows an increased amount of material. See Figure 11-10. Size and

location dimensions on the shown slot are basic. A profile tolerance of .022" is applied to the slot and referenced to three datum features. The unequally disposed symbol is shown with a value of .022" following the symbol to indicate that the entire profile tolerance applies outside the material relative to the basic profile. The alternate practice of showing the phantom line to indicate direction is



Goodheart-Willcox Publisher

Figure 11-10. A unilateral tolerance to the outside of the material (adds material) may be indicated by placing the unequally disposed symbol inside the feature control frame and repeating the tolerance value after the symbol.

also included in the figure. The phantom line is not required and is not recommended when using the unequally disposed symbol. For CAD solid models, the phantom line is not used for indicating unilateral or unequally disposed profile tolerances.

A true profile (perfect profile) sized and located by the basic dimensions forms one limit of the tolerance zone. The other limit is located .022" to the outside of the material. The effect on the shown slot is a .000" tolerance to the inside of the material and .022" tolerance to the outside. It has an effect of allowing .044" tolerance on the size (width) of the slot. The allowable variation also has an effect similar to a ± 0.011 " location tolerance for the slot center, with the magnitude of allowable location variation depending on the amount of the profile tolerance consumed by size and form variations.

An unequally disposed profile tolerance may also be applied to create a unilateral tolerance to the inside of the material. See **Figure 11-11**. Size and location dimensions in the given figure are basic. A profile tolerance of .015" is applied to the slot and referenced to three datums. The unequally disposed symbol is shown with a value of .000 (zero) following the symbol to indicate that none of the profile tolerance adds material relative to the basic profile. All of the profile tolerance applies to the inside of the material.

The effect on the shown slot is a .030" allowed increase in slot width and no allowed decrease relative to the basic dimensions. The location of the slot center relative to the referenced datums is similar to a ± 0.0075 " tolerance.

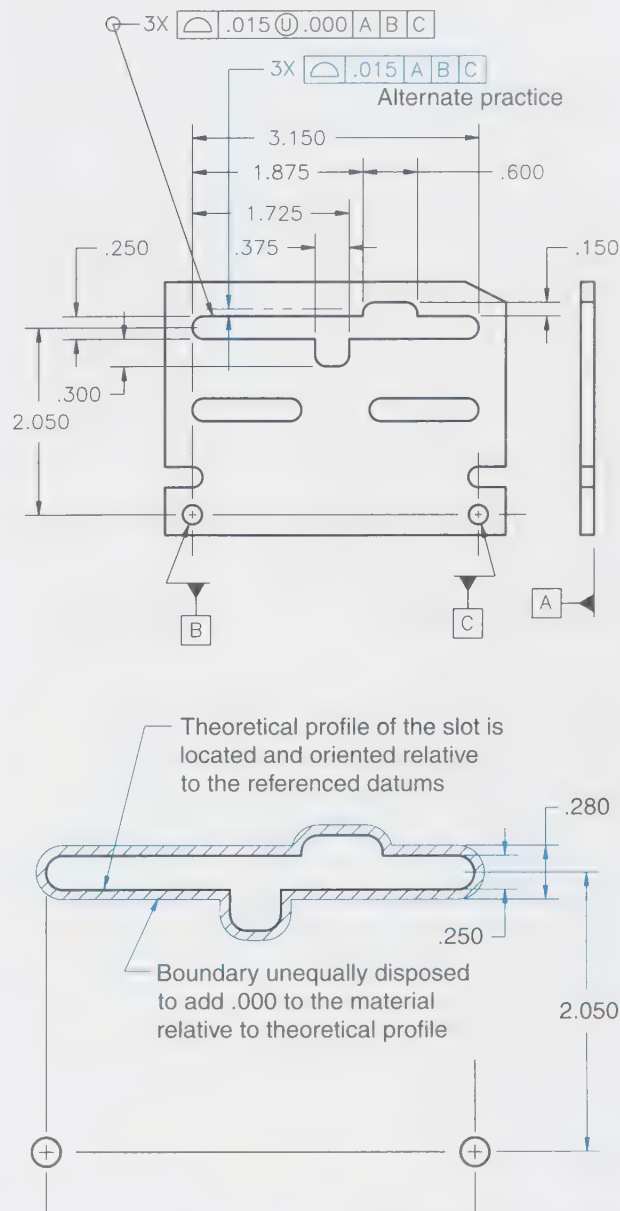
It is important to understand that a unilateral profile tolerance does not by itself establish a manufacturing target that is the true profile. It may be the design intent or desire to target the true profile, but that is not established as a requirement by applying a unilateral profile tolerance. A feature produced anywhere within the tolerance zone is equally acceptable. When using unilateral profile tolerances, it would be unwise to require a target that is the true profile if process variation is equally distributed about the fabrication target. To do so would result in a high percentage of unacceptable parts. Usually, a manufacturer will offset from the true profile when a unilateral profile tolerance is specified. The exception would be when a process has some aspect to it (such as cutting tool size) that results in variation that can only go in one direction from the true profile.

Past and Alternative Practices

Prior to the 2009 standard, the unequally disposed symbol was not used. Prior to the 2009

NOTES:

1. ALL UNTOLERANCED DIMENSIONS ARE BASIC.
2. ALL UNDIMENSIONED RADII ARE R.125



Goodheart-Willcox Publisher

Figure 11-11. A unilateral tolerance to the inside of the material may be indicated by placing the unequally disposed symbol inside the feature control frame and showing a zero value after the symbol.

standard, the means for showing a unilateral profile tolerance used a phantom line drawn on one side of the basic profile and a feature control frame connected with a leader forming a dimension across the indicated boundary. See **Figures 11-10** and **11-11**. This practice is still illustrated in the 2009 standard and may be used in addition to the unequally disposed tolerance symbol.

The phantom line is drawn on one side of the basic profile to indicate the direction in which variation is allowed. When this is done, the leader from the profile tolerance feature control frame is drawn with a segment that is perpendicular to the toleranced feature. Two arrows are applied at the end of the leader with one arrow on the phantom line and one on the basic profile.

The phantom line is only made a length that makes the direction of application obvious. It is not necessary nor is it recommended that a phantom line be shown along the full length of the toleranced feature. The distance between the phantom line and the toleranced feature must be adequate to ensure clear separation of the lines when the drawing is reproduced. The distance between the lines is not determined by the profile tolerance value.

This alternate practice is not recommended and cannot be used in CAD models.

Unequally Disposed Profile Tolerance

An unequally disposed profile tolerance is defined using the same symbol as already described for unilateral profile tolerances. See [Figure 11-12](#). The given figure shows a profile tolerance of .012" applied to the inside of the retaining ring. The tolerance includes the unequally disposed symbol followed by a value of .008". This indicates that .008" of the total .012" tolerance is in a direction that would add material. In this case, material is added to the inside of the basic profile of the feature.

This practice does not establish the true profile as a required manufacturing target. If it is

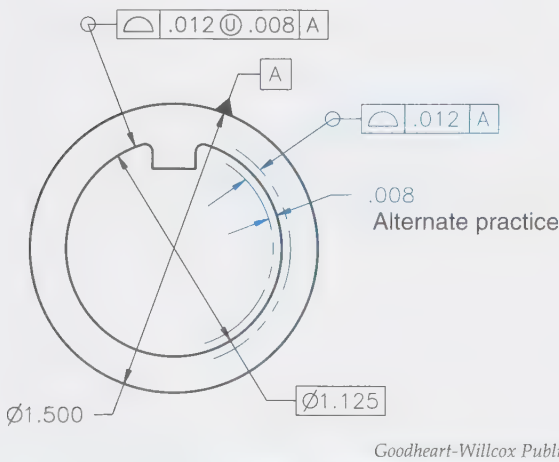


Figure 11-12. An unequally disposed tolerance zone may be indicated by placing the unequally disposed symbol in the feature control frame and following it with a value that indicates the amount of the zone that is outside the material defined by the basic profile.

desired that the true profile be the manufacturing target, then a note is required. Caution is needed when establishing a true profile as a required target, because as the amount of tolerance allocated to one side of true profile is increased, the smaller the tolerance on the other side becomes and risk of manufacturing problems increases.

The 1994 standard used phantom lines on both sides of the basic profile to show an unequally disposed profile tolerance zone. That method of showing an unequally disposed profile tolerance is still permitted as an alternate practice on orthographic views, but it is not allowed on CAD models. [Figure 11-12](#) shows the alternate practice. In the alternate practice method, a dimension is applied between the basic profile and one of the phantom lines to indicate the amount of tolerance that is applied in one direction. There is no need to use both the unequally disposed symbol and the alternate practice.

Achievable Levels of Control

Profile tolerances may be used to establish a variety of controls on part features. When profile tolerances are applied to a single surface, then it is possible to control form, a combination of form and orientation, or a combination of form, orientation, and location. If the tolerance is applied all around a closed feature such as a slot, it is possible to achieve size control in addition to form, orientation, and location.

The feature control frame is the primary factor in determining the level of control established by a profile tolerance. See [Figure 11-13](#). A profile tolerance without any datum feature reference in the feature control frame only controls form. If

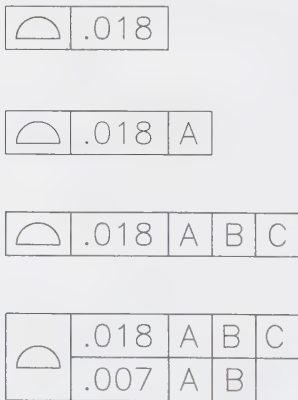


Figure 11-13. Various levels of control may be achieved through the composition of the profile tolerance feature control frame.

the feature control frame includes datum feature references, then more than form is controlled. When location dimensions are basic, the datum references result in constraining both translational and rotational degrees of freedom. If needed, the datum references may be customized (by noting applicable degrees of freedom) to constrain the rotational degrees of freedom and release the translational degrees of freedom. See Chapter 6 to review customized datum references.

Composite profile tolerances are specified using a multiple-segment (multiple-line) feature control frame that is similar to those used for composite position tolerances. Only one tolerance symbol is shown in the feature control frame. Composite profile tolerances were not illustrated in the 1982 and earlier dimensioning and tolerancing standards. To avoid possible interpretation problems when working to the 1982 or earlier standards, a note should be placed on the drawing if composite profile tolerances are utilized. The note should indicate that datum feature references in the second segment of a composite profile tolerance only constrain the rotational degrees of freedom relative to the datum reference frame.

Choices of line or surface profile tolerances combined with the available levels of tolerance specification and possible inclusion of datum feature references provide a significant amount of flexibility in what can be achieved with profile tolerances. This flexibility may be used to express the required level of tolerance control to meet functional design requirements without specifying tolerances smaller than needed. Care must be taken not to overutilize the possible controls, because overspecification of tolerances may cause product costs to increase.

Control of Surface Features

Proper application of profile tolerances is necessary to achieve the desired level of control. In addition, how feature dimensions are applied is important. It is becoming common, especially when using CAD, for all dimensions to be basic except for dimensions on simple size shapes like holes and slots. The feature shape must always be defined by basic dimensions when using a profile tolerance, but size dimensions may not be basic when the profile tolerance is applied to one surface on a feature of size. Whether or not size dimensions are basic depends on the level of control that is desired.

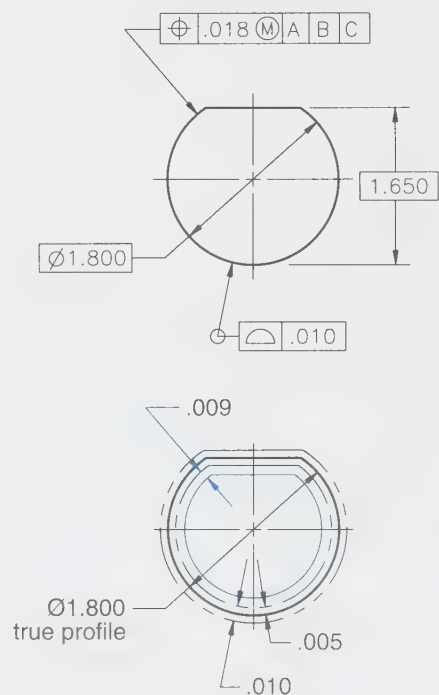
Examples of how the various levels of control may be applied using profile tolerances are provided in the following sections.

Form and Size

Form and size of an individual feature may be controlled with a profile tolerance. This should only be done when other tolerances establish the location and orientation requirements. Either profile of a line or profile of a surface may be used for specification of a form requirement, with the selection of line or surface profile depending on the design requirement. Where both form and size are to be controlled, profile of a surface is typically used. To control form only or form and size, the feature control frame for a profile tolerance must not include any datum feature reference. See [Figure 11-14](#). In the given figure, a position tolerance is used to specify the location and orientation requirement.

The given figure shows a D-shaped hole. This type of hole is commonly used in control panels in which switches and electrical connectors are mounted. The purpose for the flat side of the hole is to prevent the inserted device from rotating. It is important that the shape and size of the hole be correctly made.

Basic dimensions define the diameter of the hole and the distance across the hole at the flat. Tolerances for features with basic dimensions must be defined in a feature control frame or drawing note, because title block tolerances do not apply to basic dimensions.



Goodheart-Willcox Publisher

Figure 11-14. Control of form and size may be achieved with a profile tolerance that does not include any datum feature references.

A surface profile tolerance of .010" is applied to the hole. It includes an all around symbol, and there are no datum feature references. There is no indication of an unequally disposed tolerance zone; therefore, the tolerance zone is equal bilateral.

The tolerance zone is .010" wide and is centered on the true profile created by the basic dimensions. These tolerance zone boundaries control both the size and form of the feature. The boundary outside the material (inside the hole) is the maximum material boundary and the boundary inside the material is the least material boundary. The location of the true profile is not controlled by the profile tolerance, because the feature control frame has no datum feature references.

A position tolerance of .018" is applied at MMC relative to three datum feature references. This establishes a location requirement that is based on the maximum material boundary (MMB) created by the profile tolerance. The .018" tolerance is the total width of the tolerance zone, so the virtual condition boundary for the position tolerance is .009" inside the MMB of the feature.

Form, Orientation, and Location

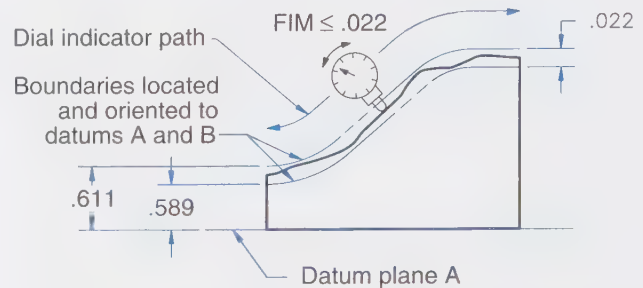
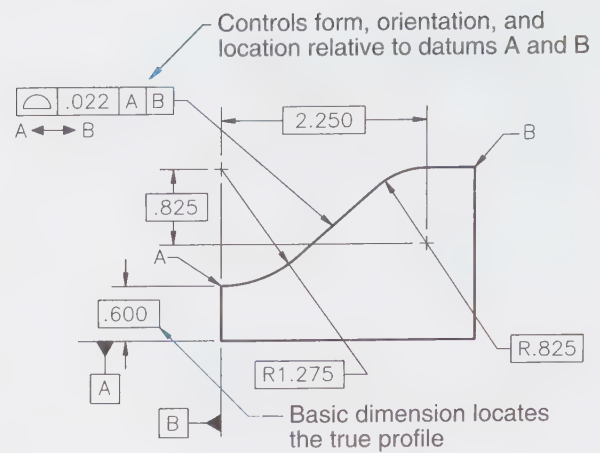
Profile tolerance specifications that include datum feature references control form, orientation, and location when the tolerated feature is located by basic dimensions. See Figure 11-15. This figure shows a surface profile tolerance of .022" applied to the upper surface. The surface is defined by basic dimensions including the .600" location dimension from datum feature A. The location from datum feature B is also basic at a value of .000" from the datum feature to the center of the 1.275" radius.

The tolerance zone on the given surface is .022" wide. It is centered on the true profile created by the basic dimensions that locate and define the shape of the surface. The tolerance zone is constrained in translation and rotation relative to the referenced datums, resulting in a tolerance zone that controls the location and orientation of the feature surface.

The tolerance zone is centered on the true profile because there is no indication of an unequally disposed tolerance. The profile variation is measured normal to the basic profile.

Controls on Coplanar Features

Large machined parts sometimes have raised areas, called *bosses*, machined to create localized mounting pads. This prevents surface variation across large areas from causing distortion when the part is bolted to another part. The same technique is used on castings and forgings. Local



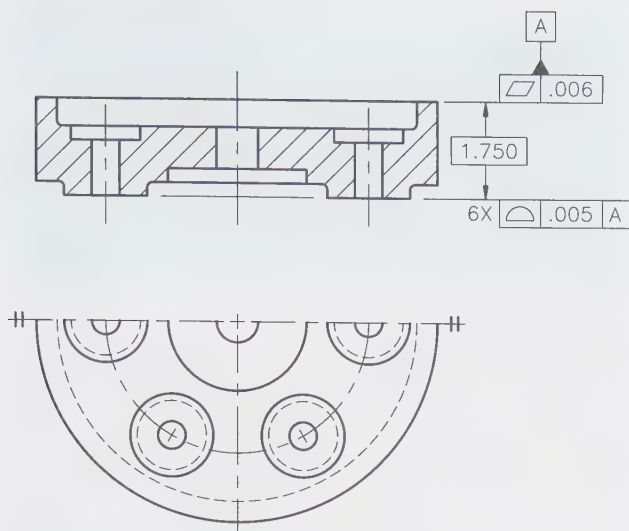
Goodheart-Willcox Publisher

Figure 11-15. Application of basic location dimensions and datum feature references in the profile tolerance constrains the translational and rotational degrees of freedom of the tolerance zone.

bosses are built into the cast or forged parts and later machined to create coplanar surfaces that provide good mounting provisions.

Specification of the coplanarity requirements for multiple features is achieved through the specification of profile tolerances. See Figure 11-16. The level of control achieved may be varied by what is shown in the dimensions and tolerance specification. All the information shown in the given figure may be used to achieve the maximum level of control. Some or all of the shown tolerance specification and dimensions may be eliminated, depending on the desired control.

Flatness tolerances are not used to control coplanarity of features such as the bosses in Figure 11-16. Flatness is a control of individual features. Specification of a flatness tolerance on six bosses would permit production of six flat bosses, no two of which lie in a common plane. It would not be a good practice to attempt to achieve coplanarity by combining flatness and the continuous feature (CF) symbol on the bosses. A flatness tolerance in combination with a continuous feature symbol is subject to interpretation differences because this combined utilization of symbols is not defined by the current dimensioning and



Goodheart-Willcox Publisher

Figure 11-16. The level of control on coplanar surfaces is determined by the size dimensions and profile tolerance specifications placed on the part.

tolerancing standard. Profile has a well-defined meaning when applied to establish coplanarity; therefore, profile is the correct tolerance characteristic to specify for this purpose.

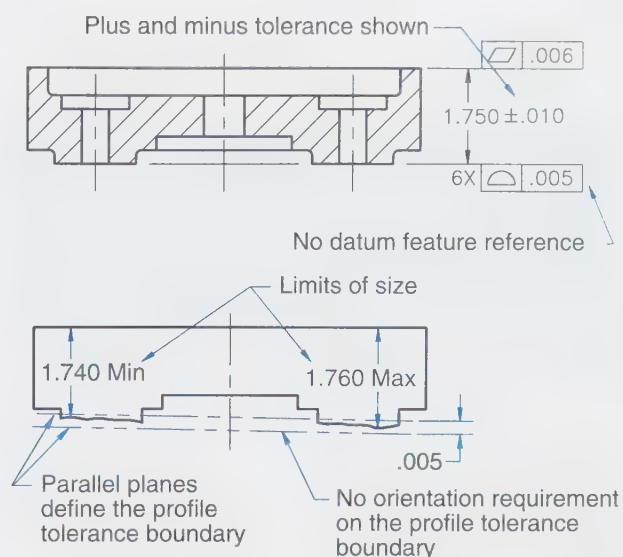
Coplanarity Only

The amount of control placed on a feature or group of features should reflect the functional requirements of the part. In some designs, coplanarity of features is important to create a good mounting plane for a part. It is possible that the location and orientation of the mounting plane is relatively unimportant. For this type of situation, a profile tolerance may be applied to control only coplanarity.

Figure 11-17 shows the section view from **Figure 11-16**. This figure shows a surface profile tolerance of .005" applied to the six bosses on the part. The profile tolerance specification is on an extension line from the bosses and defines a requirement that is applicable to all six bosses.

A $1.750 \pm .010$ " dimension is applied to the thickness of the part. This is a size dimension and will, based on Rule #1, create a perfect form boundary. It will thereby indirectly control the orientation of each boss relative to the opposite surface.

The combination of the $\pm .010$ " size tolerance and the .005" profile tolerance defines the coplanarity, location, and orientation requirements. Coplanarity is controlled by the .005" profile tolerance. The profile tolerance zone may be in any location and orientation, but the surfaces of all six bosses must lie between two planes that are separated by .005".



Goodheart-Willcox Publisher

Figure 11-17. Coplanarity is required by the profile tolerance if no datum feature references are shown.

Location and orientation of the surfaces on the bosses are controlled by the $\pm .010$ " tolerance on the size dimension. It is possible for the lowest point on one of the bosses to be at 1.740", and the highest point on another boss to be at 1.760". This is permitted so long as all surfaces are contained within two planes separated by .005" as required by the profile tolerance.

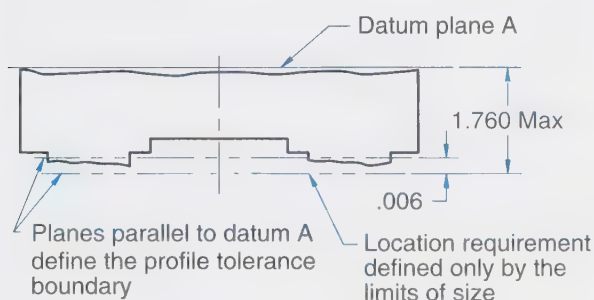
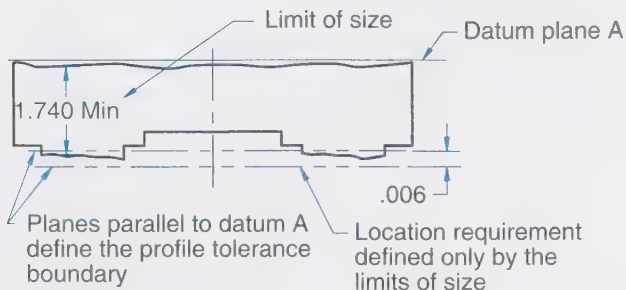
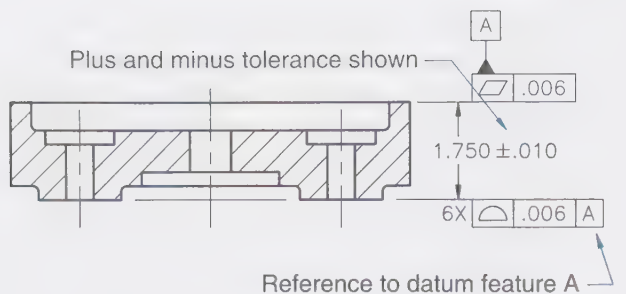
The same level of tolerance control may be accomplished using composite profile tolerances as explained later in this chapter. The utilization of composite profile tolerances is recommended.

Coplanarity and Orientation

Orientation is sometimes as important as the required coplanarity for the mounting surfaces. A combined coplanarity and orientation requirement may be specified using a profile tolerance that references a datum feature. See **Figure 11-18**.

The previously shown figure is repeated with some changes to the dimensioning and tolerancing. A datum feature symbol has been added to one surface. The surface profile tolerance applied to the bosses is .006" and is referenced to datum feature A. On the given part, there is no need to show a basic angle between the datum feature and the controlled surfaces because drawing convention permits parallelism to be assumed in orthographic views.

The profile tolerance zone is constrained in rotation relative to the referenced datum at the angle shown on the drawing. In this case, it is parallel to datum A. The profile tolerance zone requires coplanarity within a .006" zone, and also requires parallelism within the same zone.



Goodheart-Willcox Publisher

Figure 11-18. Coplanarity and orientation of the tolerance zone are required if a datum feature reference is shown. The tolerated size dimension controls the distance between the datum feature and the top of the bosses.

Location of the bosses is allowed to vary within the $\pm .010$ " tolerance on the 1.750" size dimension. The $\pm .010$ " tolerance is not reduced by the profile tolerance, but is refined by the allowable orientation variation invoked by the profile tolerance. If a part is produced with the lowest point on one of the bosses at 1.740", then the highest point on any other boss is controlled by the .006" profile tolerance.

Another part may be produced with the highest point on one boss at 1.760". If this occurs, the lowest point on any other boss is controlled by the .006" profile tolerance.

The profile tolerance does, in effect, control size variation within a .006" tolerance zone that is parallel to datum A, but the .006" zone may fall anywhere within the $1.750 \pm .010$ " tolerance zone.

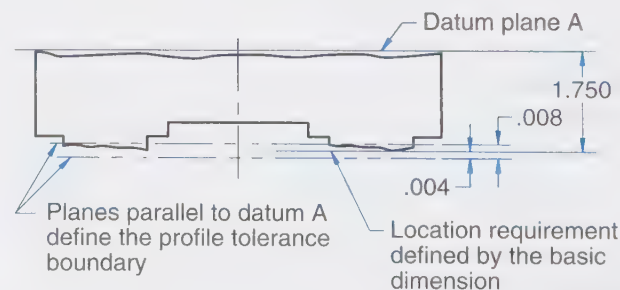
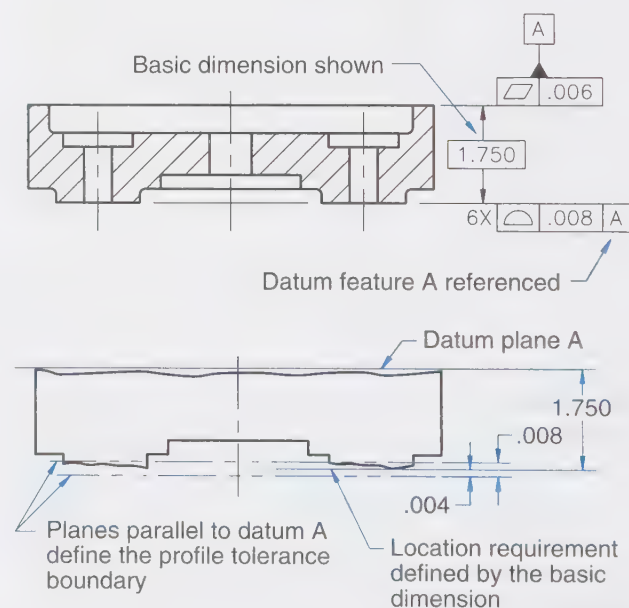
Pro Tip

The same level of tolerance control may be accomplished using composite profile tolerances as explained later in this chapter. The utilization of composite profile tolerances is recommended for achieving this level of tolerance control.

Coplanarity, Orientation, and Location

When the location of a group of features is as important as the orientation and coplanarity, all three parameters may be controlled through the specification of one profile tolerance and application of basic dimensions. See **Figure 11-19**. The previously shown figure is repeated and the necessary changes have been made to result in the profile tolerance controlling coplanarity, orientation, and location. The drawing includes one surface identified as datum feature A. A surface profile tolerance of .008" referenced to datum feature A is applied to the bosses. A basic dimension of 1.750" is used to locate the bosses.

Profile tolerances control location of a feature if the feature is located with a basic dimension. A theoretically true profile of the bosses is established at the 1.750" dimension. The profile tolerance specification establishes a .008" wide tolerance zone bounded by two parallel planes. This tolerance zone is centered on the 1.750" location.



Goodheart-Willcox Publisher

Figure 11-19. Coplanarity, orientation, and location are controlled by the profile tolerance if a datum feature reference is shown and the size dimension is basic.

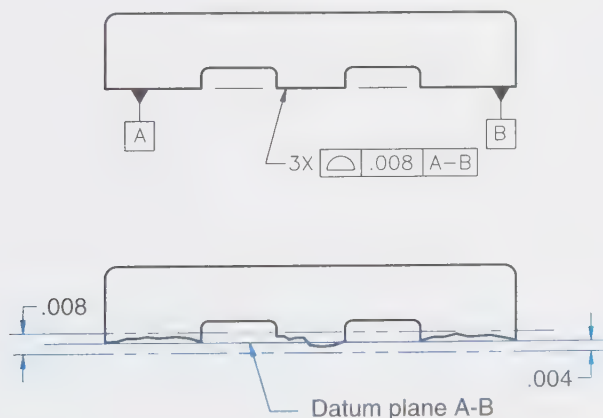
Surface variations on the bosses must lie between the two planes that form the tolerance zone. Surfaces that lie within the zone are coplanar within .008", oriented to datum A within .008", and located within a .008" zone that is centered on the 1.750" basic dimension.

Coplanarity Relative to a Datum Plane Established by the Controlled Features

Some controversy existed prior to the 1994 standard regarding the requirements created when a profile tolerance is applied to multiple features and referenced to a plane established by one or more of the controlled features. There is now a well-defined meaning in ASME Y14.5-2009. See **Figure 11-20**. The given figure shows a part with three lands on it. Two of the lands are identified as datum features A and B. A profile tolerance of .008" is applied to all three surfaces, and referenced to datum feature A-B.

Datum plane A-B is established from the surfaces of features A and B. The profile tolerance zone is bilateral and is therefore centered on the datum plane. This creates a .008" wide tolerance zone for all features other than the datum features. The datum features can only use half the specified tolerance zone because it is impossible for them to be located below the datum plane.

If this effect is undesirable, a profile tolerance without the datum references could be applied to the three lands. That will permit a full .008" variation on all the surfaces. Another way to create a tolerance zone that is .008" for all of the features including the datum features is to use an unequally disposed tolerance zone with a .000" tolerance to the outside of the material. These two options result in a different amount of allowed variation when



Goodheart-Willcox Publisher

Figure 11-20. A profile tolerance may be referenced to a datum plane that is established by the controlled feature(s).

compared to the applied tolerance in the figure. They will also affect the location of datum A.

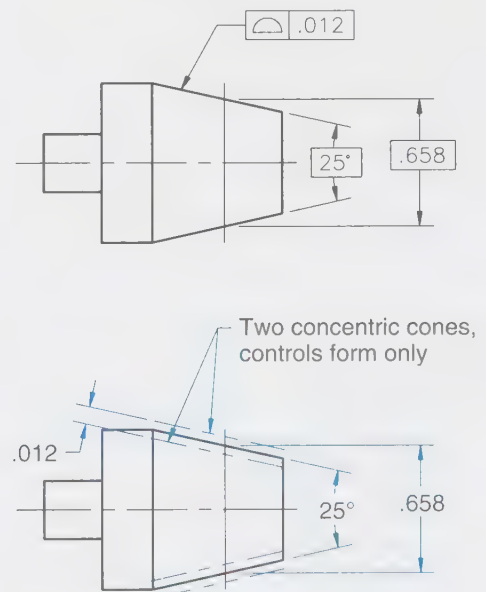
Note

Do not use an unequally disposed zone that puts all the tolerance to the outside of the material on this part. That would result in zero available tolerance relative to the datum and that cannot be achieved.

Profile Applied to Conical Surfaces

Profile tolerances provide a means for controlling the form, orientation, and location of conical surfaces. No other tolerance permits control of these parameters for a conical feature. Circular runout may be used to control circular elements on a cone, but that does not control the overall form of the cone. Some control of form on a cone may be achieved with straightness tolerance along the length and circularity tolerance for each cross section.

Control of the overall cone form is specified with a surface profile tolerance. If no datum feature references are included, the tolerance controls only form. See **Figure 11-21**. In the given figure, all dimensions that define the cone are basic. A surface profile tolerance of .012" is applied to the cone surface with a leader. Placement on an extension line from the surface is also permitted.



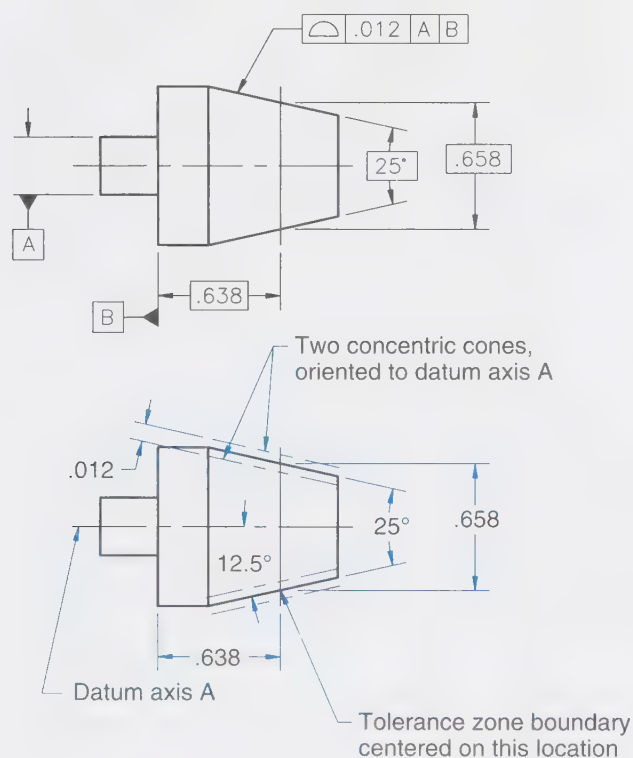
Additional requirements needed to establish location, orientation, and size

Goodheart-Willcox Publisher

Figure 11-21. The form of a conical surface may be controlled through the use of a profile tolerance.

Surface variations on the cone must fall within two concentric cones that are separated by the specified .012" tolerance. The two concentric cones control only form even though the cone dimensions are basic and the tolerance zone is centered on the given dimensions. Although the two cones forming the tolerance zone boundary must remain concentric, they can move together to any orientation or location. This is allowable because no datum feature references are shown in the tolerance specification. This type of specification leaves questions open regarding the location and orientation of the cone. Additional geometric controls should be used to establish location and orientation requirements. That may be done through the application of basic dimensions and by adding datum feature references to the profile tolerance. It may also be possible to use a composite profile tolerance if allowable location tolerances are larger than the needed form tolerances.

Form, orientation, and location of a conical surface may be controlled with a profile tolerance specification that includes a datum feature reference. See **Figure 11-22**. The previously shown figure is repeated, but references to datum features A primary and B secondary have been added to the



Goodheart-Willcox Publisher

Figure 11-22. Orientation and location of a conical surface may be controlled by a profile tolerance including appropriate datum feature references and using basic dimensions to locate the surface.

profile tolerance specification. A basic dimension locating the conical feature relative to datum B was added.

Referencing datum features A and B requires that the degrees of freedom for the tolerance zone be constrained in translation and rotation relative to the datum reference frame. Surface variations may vary in location and orientation provided they fall within the tolerance zone.

Composite Profile Tolerances

Design applications often require tighter tolerances on the form of a feature than the tolerances required for the orientation or location of the same feature. One example is the mounting surfaces for a piece of optical equipment. The coplanarity of the surfaces is very important to prevent distortion of the optics when the parts are fastened in place. Orientation and location of the mounting surfaces may be less important because adjustment provisions may be included in the design of the optical equipment. The adjustments provide a means of compensation for orientation and location variations.

Datum references in a composite profile tolerance result in the same requirements relative to the datum reference frame as are required for position tolerances. Datum references in the first segment constrain the translational and rotational degrees of freedom relative to those datums. Datum references in the second or lower segments constrain only the rotational degrees of freedom.

Composite profile tolerances permit specification of relatively large tolerances for location, orientation, and form, with a second requirement that more closely controls orientation and form or in some cases only form. See **Figure 11-23**. The given part has two lands that have a composite profile tolerance applied to them.

The first segment of the composite tolerance references datum feature A, so it creates a tolerance zone .020" wide that has the translational and rotational degrees of freedom constrained relative to datum A. The tolerance zone is bounded by two planes that are parallel to datum A. The tolerance zone is centered on the true profile that is located by the 1.000 basic dimension.

The second segment of the composite profile tolerance establishes a coplanarity requirement that is bounded by two parallel planes separated .007". Because there are no datum references in the second segment, it establishes no requirement to constrain the rotational degrees of freedom for the tolerance zone. The tolerance zone may move to best fit on the varied surfaces and will control

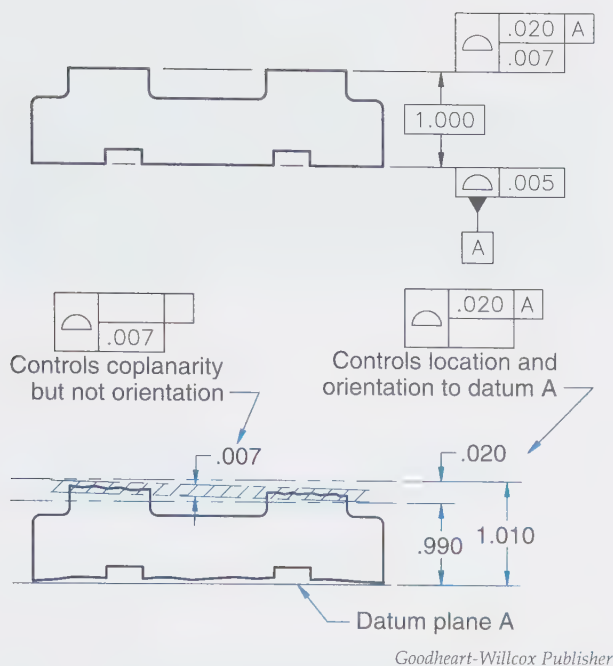


Figure 11-23. Composite profile tolerances permit large tolerances for less critical characteristics, and close control of the characteristics that need smaller tolerances.

coplanarity and form. Although the tolerance zone created by the second segment may float, the surface variations on the part must fall within the requirements of both the first and second segment tolerance zones.

A composite profile tolerance may include a second segment that does constrain one or more of the rotational degrees of freedom. This requires datum feature references in the second line of the composite tolerance. See [Figure 11-24](#). The reference to datum feature A in the second segment requires the tolerance zone to be constrained in the rotational degrees of freedom relative to datum A, but does not require translation to be constrained. This will have the impact of controlling the surfaces to be coplanar and parallel to datum A within the .007" tolerance. The location of the surface is free to vary within the tolerance of the first segment.

Composite Profile All Around

Composite profile may be applied to enclosed features with the tolerance applied all around. See [Figure 11-25](#). The five-sided hole through the part has a composite profile tolerance applied all around. The first segment requires a profile tolerance that is .080" wide and the translational and rotational degrees of freedom are constrained relative to the datum reference frame A primary, B secondary, C tertiary. The second segment creates a

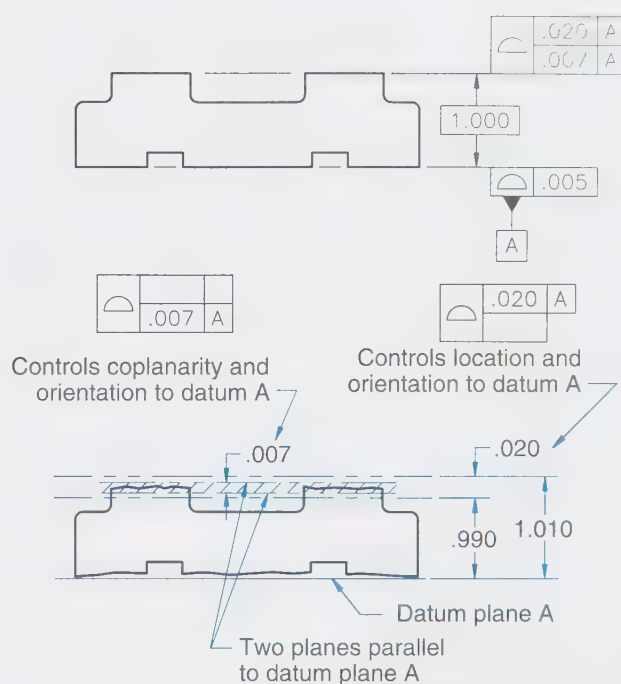
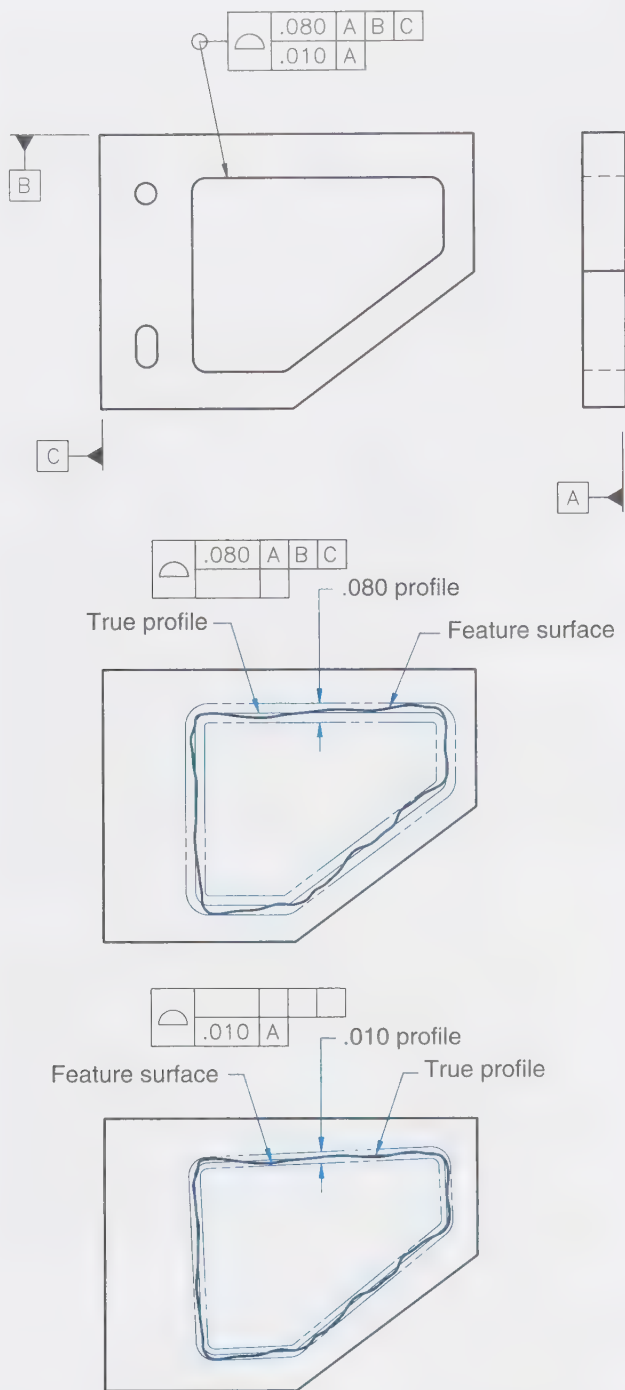


Figure 11-24. Datum feature references in the second line of a composite profile tolerance may be used to constrain the rotational degrees of freedom of the tolerance zone relative to the datum reference frame.

tolerance zone that is .010" wide and only the rotational degrees of freedom relative to datum A are constrained. This second segment controls the size and form of the feature as well as the orientation to datum A. No rotational constraint is established relative to datums B or C, so the tolerance zone is only required to be perpendicular to datum A. The tolerance zone created by the second segment is free to translate relative to the datums, but the surface must be within the .080" wide zone created by the first segment.

Profile and Position Tolerances Combined

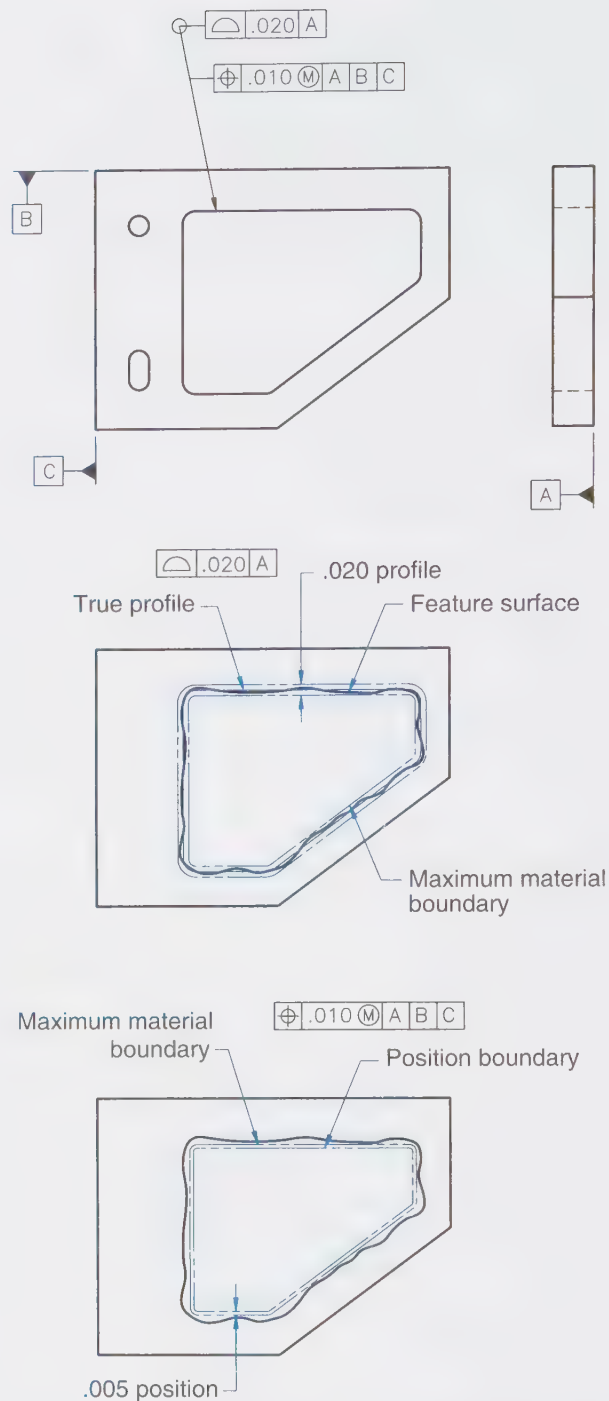
A profile tolerance may be used to establish size, form, and orientation requirements and a position tolerance applied to control location. See [Figure 11-26](#). The profile tolerance creates two boundaries for the controlled features. One of those boundaries is the maximum material condition for the feature. The position tolerance applied includes the MMC modifier, indicating the tolerance is applicable to the MMC of the controlled feature. The position tolerance value is the total difference between the MMC size and the position tolerance boundary. In the given part, the .010" tolerance results in a boundary that is .005" inside the MMC boundary.



Goodheart-Willcox Publisher

Figure 11-25. Composite profile may be applied to a feature so that the feature-relating tolerance (second segment) controls size, form, and orientation to the primary datum.

This is not the same as a composite profile tolerance. One difference is that the composite profile must permit a larger tolerance in the first segment (location tolerance) than is given in the second segment. Profile combined with position tolerances permits the profile tolerance to be larger than the position tolerance because the position toler-



Goodheart-Willcox Publisher

Figure 11-26. In this example, a profile tolerance controls size, form, and orientation, and a position tolerance controls location.

ance is controlling the location of the maximum material boundary. Another difference occurs when the position tolerance includes the MMC modifier. It creates a virtual condition boundary that is fixed in size. As the feature departs from the maximum material boundary, the feature may have additional position variation and not violate the virtual condition boundary. In effect, there is

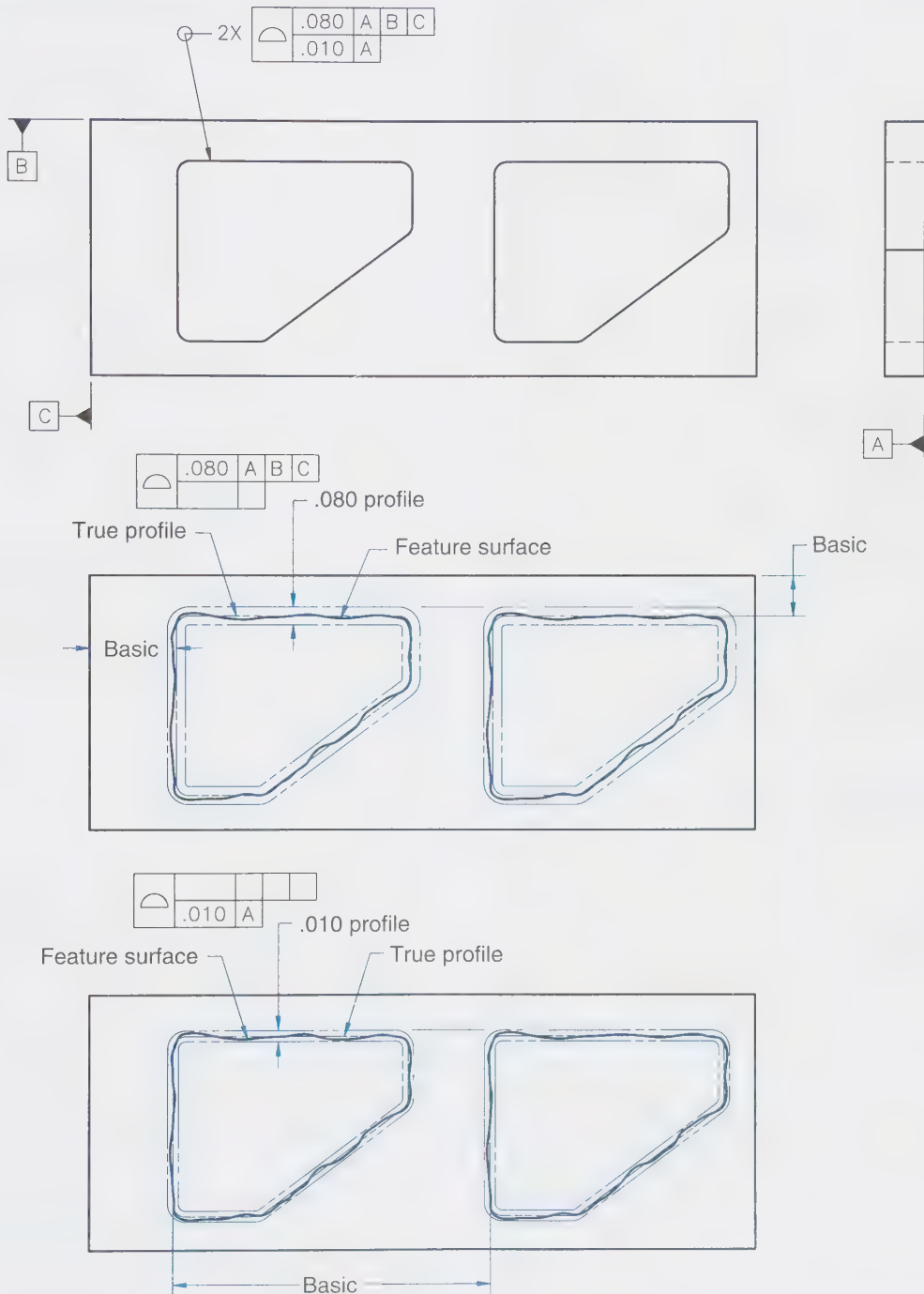
a bonus tolerance just as when position tolerances are applied to simple features of size such as holes.

Pro Tip

It is not recommended that a position tolerance be specified RFS on a feature that has a profile tolerance applied to it.

Composite Profile for Pattern of Features

Multiple features that form a single pattern may be tolerated with a composite profile tolerance. See **Figure 11-27**. The first segment creates a pattern-locating tolerance zone framework that is made up of the profile tolerance zones around each of the features. The pattern-locating tolerance zone framework has all possible translational and



Goodheart-Willcox Publisher

Figure 11-27. Composite profile may be applied to a pattern of features so that the feature-relating tolerance (second segment) controls size, form, and orientation to the referenced datums and also controls the feature locations relative to one another.

rotational degrees of freedom constrained relative to the referenced datums. In the given figure, the pattern-locating tolerance zones are .080" wide and are bilateral. The surfaces of the tolerated features must be within the pattern-locating profile tolerance zones.

The second segment defines the feature-relating tolerance zone framework. The feature-relating tolerance zones for the features are basically located relative to one another and the entire framework has the rotational degrees of freedom constrained relative to the referenced datum. This, in effect, controls the spacing between the features as well as the orientation and form of the features. In this case, the .010" wide feature-relating tolerance zones are constrained in rotation relative to datum A. The surface of the features must be within the feature-relating tolerance zones. The feature-relating tolerance zone framework is free to rotate relative to datums B and C provided the feature surfaces stay within both the pattern-locating and the feature-relating tolerance zones.

It is possible to include a reference to datum feature B in the second segment. The reference to datum feature B in the second segment would constrain the rotational degree of freedom for the tolerance zone framework relative to datum B.

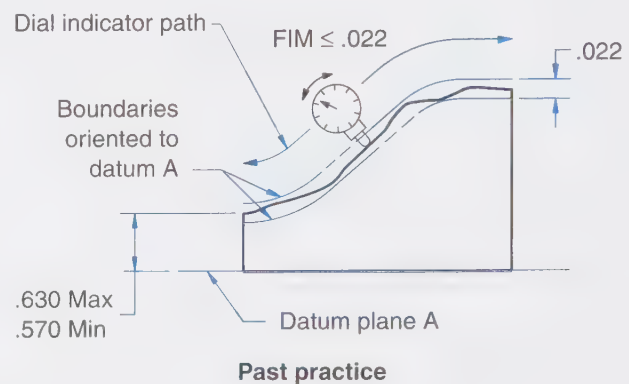
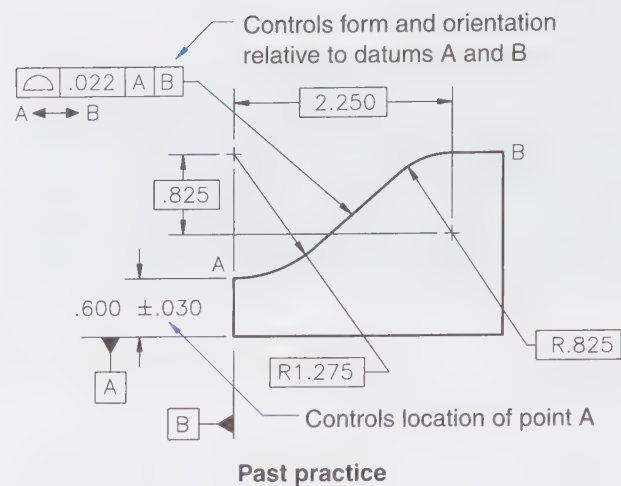
If there is a need to more tightly control the form of individual features within the pattern, another single-segment feature control frame could be included. Placing a notation of INDIVIDUALLY adjacent to that feature control frame would indicate the control is applied to each feature independently.

Past Practices

This explanation is retained for work that must be done with drawings created in the past. The practice described is not supported by the current standard and was not completely defined in past standards. This practice can be better achieved with composite profile tolerances as previously explained and should not be used in new product documentation.

Past practice permitted the application of coordinate tolerances to locate surfaces. Profile tolerances could be used in conjunction with coordinate tolerances to control form and orientation relative to one or more datums. See **Figure 11-28**. Although the practice was permitted, the interpretation was subject to debate and more than one possible interpretation was possible. The following is one interpretation.

This method does not use basic dimensions to define location of the surface. A height dimension



Goodheart-Willcox Publisher

Figure 11-28. This past practice was not well defined in previous standards. The current standard provides a better means of achieving the desired tolerances using composite profile feature control frames.

including a coordinate tolerance of $\pm.030$ " is used to locate the curved surface relative to the bottom of the part.

A surface profile tolerance specification of .022" is attached to the surface. It includes a reference to datum feature A. The tolerance zone established by the specification is .022" wide, and it is constrained in rotational degrees of freedom relative to the referenced datum. The location of the tolerance zone may float vertically within the $\pm.030$ tolerance on the .600 dimension, provided the orientation to datum A is maintained.

Location of the surface is controlled by the $\pm.030$ " tolerance that locates one corner. The corner may be anywhere within the .060" wide location tolerance band. Surface variations may go in any direction from the located corner, provided the .022" profile tolerance zone may be moved into a position that contains all the surface variations. The profile tolerance zone may be moved to any position, provided the orientation to datum A is maintained.

Chapter Summary

- ✓ Profile of a line controls line elements that are parallel to the profile on which the tolerance is specified.
- ✓ Profile of a surface simultaneously controls all points on a surface.
- ✓ Profile tolerances establish requirements for the entire surface to which they are applied unless limits of application are defined.
- ✓ An all around symbol is required if a profile tolerance is to apply beyond the abrupt changes in direction that define the boundary of a feature.
- ✓ The all over symbol is used when the profile tolerance is applicable to all surfaces.
- ✓ Tolerance zones created by specified profile tolerances are equally disposed bilateral unless the unequally disposed profile symbol is used to indicate otherwise.
- ✓ An unequally disposed profile symbol may be used to indicate that a profile tolerance applies in one direction relative to the profile defined by basic dimensions.
- ✓ Levels of control that may be achieved with surface profile tolerances include form, form and orientation, and form, orientation, and location. Form control also includes size control in some situations.
- ✓ Coplanarity requirements may be specified with a profile tolerance.
- ✓ Coplanarity, orientation, and location of multiple features may be achieved with profile tolerances. The level of control depends on how the part is dimensioned and whether or not datum feature references are used.
- ✓ Profile is the only geometric tolerance that may be used to specify size, form, orientation, and location requirements for a cone.
- ✓ Composite profile specifications may be used to indicate a relatively large tolerance to define location requirements and a smaller tolerance for requirements such as form and orientation.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

1. Profile tolerance specifications may include ____ datum feature references.
 - A. no
 - B. one
 - C. two or three
 - D. All of the above.
2. Profile tolerance values are assumed to apply ____.
 - A. RFS
 - B. MMC
 - C. LMC
 - D. None of the above.
3. A profile tolerance with ____ datum feature references only controls form or possibly form and size.
 - A. no
 - B. one
 - C. two or three
 - D. None of the above.
4. Variations measured on one surface element are independent of the variations on adjacent elements on the same surface if a ____ tolerance is used.
 - A. flatness
 - B. surface profile
 - C. line profile
 - D. composite profile
5. When a profile tolerance is applied to a portion of a feature, ____ must be indicated.
 - A. datums
 - B. a surface profile tolerance
 - C. a line profile tolerance
 - D. limits of application
6. Showing a datum feature reference in the second segment of a composite profile tolerance specification establishes that the ____ degrees of freedom for the tolerance zone must be constrained relative to the datum reference frame.
 - A. rotational
 - B. translational
 - C. Always A and B.
 - D. Neither A nor B.
7. Coplanarity of flat surfaces should be achieved with a ____ tolerance.
 - A. size
 - B. surface profile
 - C. line profile
 - D. flatness

8. Profile tolerances may be applied to control the _____ of a conical surface.
A. angle
B. size
C. location
D. All of the above.

True/False

9. *True or False?* References to datum features of size in a profile tolerance specification are assumed to apply RMB, but MMB modifiers are permitted.
10. *True or False?* In one respect, line profile is similar to a straightness tolerance applied to a surface, because both straightness and line profile tolerances control individual surface elements.
11. *True or False?* Unless otherwise specified, all profile tolerances are assumed to be totally distributed on one side of a theoretically perfect shape.
12. *True or False?* The combination of basic dimensions used to locate a feature and profile tolerances referenced to datum features creates a tolerance zone that has translational and rotational degrees of freedom constrained relative to the datum reference frame.
13. *True or False?* Profile tolerances may be applied to only permit variation to one side—inside or outside—of the basic feature outline.

Fill in the Blank

14. Two types of profile tolerances are _____ and _____.
15. The shape of a curved feature must be defined by _____ dimensions if a profile tolerance is applied to the surface.
16. Surface profile establishes a tolerance zone across _____ of a surface.

17. A profile tolerance extends across a feature to _____ changes in surface direction.
18. A(n) _____ disposed symbol shown in a feature control frame and followed by a number indicates the amount of the profile tolerance that is to extend outside the material defined by basic dimensions.
19. The _____ segment of a composite profile tolerance constrains only rotational degrees of freedom relative to any referenced datums.

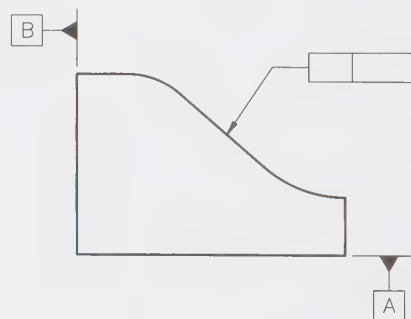
Short Answer

20. Show the two profile tolerance symbols and label each symbol.
21. Explain how a profile tolerance may be specified to extend all the way around a feature.
22. When a composite profile tolerance is applied to control a pattern of features, what requirement does the first segment of the tolerance specification establish?

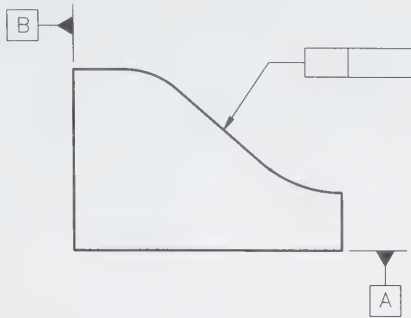
Application Problems

Each of the following problems requires that a sketch be made of the figure. All sketches should be neat and accurate. Apply all required dimensions in compliance with dimensioning and tolerancing requirements. Show any required calculations.

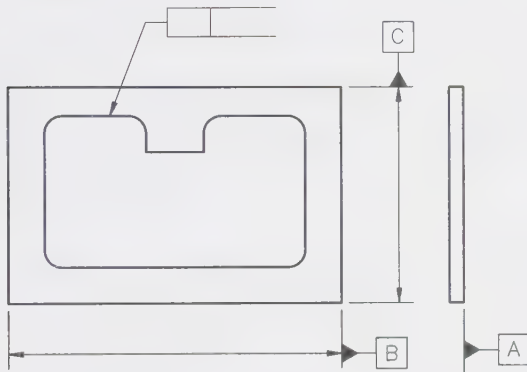
23. Apply a bilateral surface profile tolerance of .020" relative to datum feature A primary and datum feature B secondary.



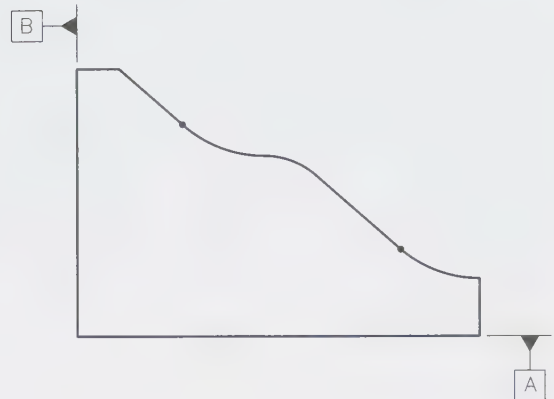
24. Apply an unequally disposed surface profile tolerance of .015" with the full .015" zone lying to the outside of the material. Reference datum feature A primary and datum feature B secondary. Use the unequally disposed symbol and do not show the alternate method.



25. Apply a surface profile tolerance of .018" all around the cutout. Relate the tolerance to datum features A primary, B secondary, and C tertiary. Datum features B and C should be RMB.



26. Apply a surface profile tolerance of .030" relative to datum feature A primary and datum feature B secondary. Limit the application to the surface area between the indicated points. Label the points.

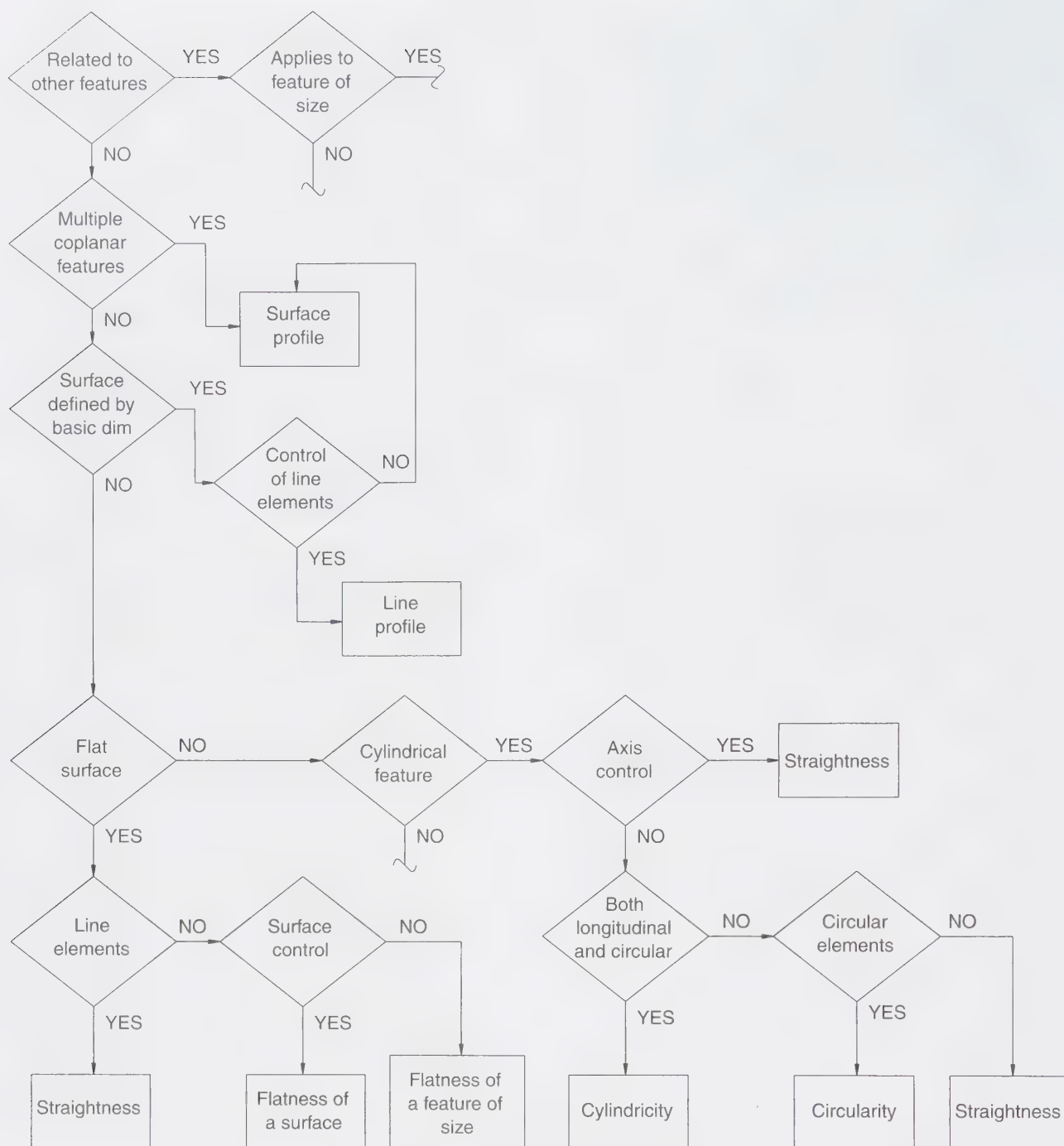


27. Use a composite profile tolerance to control the three surfaces to be coplanar and oriented to datum feature A within .008" and located to datum feature A within .020".



28. Control the three surfaces to be located, oriented, and coplanar within .012" relative to datum feature A.





Goodheart-Willcox Publisher

Chapter 12

Practical Applications and Calculation Methods

Objectives

Information in this chapter will enable you to:

- ▼ Calculate position tolerances when more than two parts are stacked in a floating fastener or fixed fastener application.
- ▼ Distribute the total available position tolerance between features to which position tolerances are applied.
- ▼ Specify projected tolerance zones for fixed feature locations to prevent interference conditions.
- ▼ Determine the amount of tolerance accumulation in a simple assembly.
- ▼ Properly use zero position tolerances at MMC to increase manufacturing freedom.
- ▼ Apply paper gaging techniques to determine if a produced part meets drawing requirements.
- ▼ Explain allowable position tolerance effects resulting from datum references at MMB.

Technical Terms

partially fixed fastener
projected tolerance zone
zero tolerancing at MMC

Introduction

Floating fastener and fixed fastener conditions include two or more parts that are assembled together. Regardless of the number of parts in an assembly, tolerances can be calculated to ensure proper assembly. This chapter shows how to calculate position tolerances for assemblies containing two or more parts stacked together. It also shows how to calculate tolerances when multiple parts

are assembled in a manner that results in an accumulation of tolerances.

Application of the correct tolerance to properly specify the needed level of control requires that careful consideration be given to the design application. A logical process can be used to determine the proper type of tolerance, or groups of tolerances, to apply.

Application of tolerance zones that specify the required level of accuracy and permit the maximum manufacturing flexibility requires an understanding of how the tolerances affect the part. It also requires that calculations be completed on the basis of how the parts fit together. A proper understanding of tolerance types and concepts such as zero tolerancing at MMC will ensure tolerances are maximized.

Care must be exercised in the application of position tolerances at MMC and datum references at MMB. The additional tolerance resulting from the modifiers can cause mixed results. A benefit is that achieving the hole position tolerance on a detail part becomes easier. A negative is that assembly variation can increase as a result of the additional tolerance on the detail parts.

Floating Fastener Condition

The floating fastener condition was described to some extent in previous chapters. A more in-depth look at the floating fastener condition makes it possible to see how calculations of tolerances can be completed for simple or complex designs that include floating fasteners.

A floating fastener condition exists any time a bolt, pin, shaft, or other feature passes through clearance holes in an assembly. A floating fastener condition does not exist if any of the holes contain

threads, a press fit, or any other feature that fixes the location of the fastener. The fastener must be free to float within all the holes.

It is important to know that clearance holes in parts do not always act as floating fastener conditions. If the amount of position tolerance or orientation tolerance applied to a hole is greater than the amount of clearance that exists between the hole and the fastener that passes through the hole, the hole can affect the fastener location in the assembly.

The fastener location is affected by a produced hole any time the hole location variation is greater than the clearance between the hole and the fastener. This can be called a *partially fixed fastener* condition and it must be taken into consideration when the total tolerance is distributed between parts in an assembly.

It is also important to note that position tolerance calculations shown in this chapter are based on the interactions between the hole location tolerances and the fastener sizes. Other factors that may impact the fit of the fastener as it passes through the holes are not included. One example of such a factor is the parallelism of the surfaces on the parts. If a flat plate has parallelism variation, that variation can have a negative impact on the alignment of holes. The effect of geometric variations on the fit of parts is an extensive subject that is covered in texts on variation analysis and dimensional management.

One factor that can benefit fastener installation is that a fastener may pass through two misaligned holes in mating parts if the fastener is inserted at a slight angle. On relatively thin parts, this will allow a fastener to pass through a pair of holes even though they appear to create a shape that has a minimum diameter slightly smaller than the fastener. The result will be a fastener head that does not sit flat on the part surface prior to being tightened. This is not always an acceptable practice because it can impact strength and fatigue life. It is usually an engineering decision whether any gap is permitted under the fastener head.

The following explanations are based on the fastener not being allowed to install at an angle; thus, the fastener head will sit flat. The explanations also assume the parts are rigid, and that the fastener is not permitted to deform the hole. The calculations in this chapter accommodate the worst case condition of holes being located in extreme locations opposite one another. Statistically, worst case is very unlikely to ever occur, so the calculations are conservative.

Two Stacked Parts

The simplest assembly in which a floating fastener condition exists is one in which two parts are stacked together. See **Figure 12-1**. Each of the given parts is a simple rectangular plate with two clearance holes. Only one of the holes is shown in the given section view. Bolts pass through the clearance holes.

An assumed functional requirement for the assembled parts is to have two edges aligned. These edges are identified as datum features, and dimensions to locate the holes originate from these datum features. Position tolerances specified for the holes reference the datum features. A single-segment feature control frame for position tolerance is used to ensure the edges of the parts will align and the bolts pass through the holes.

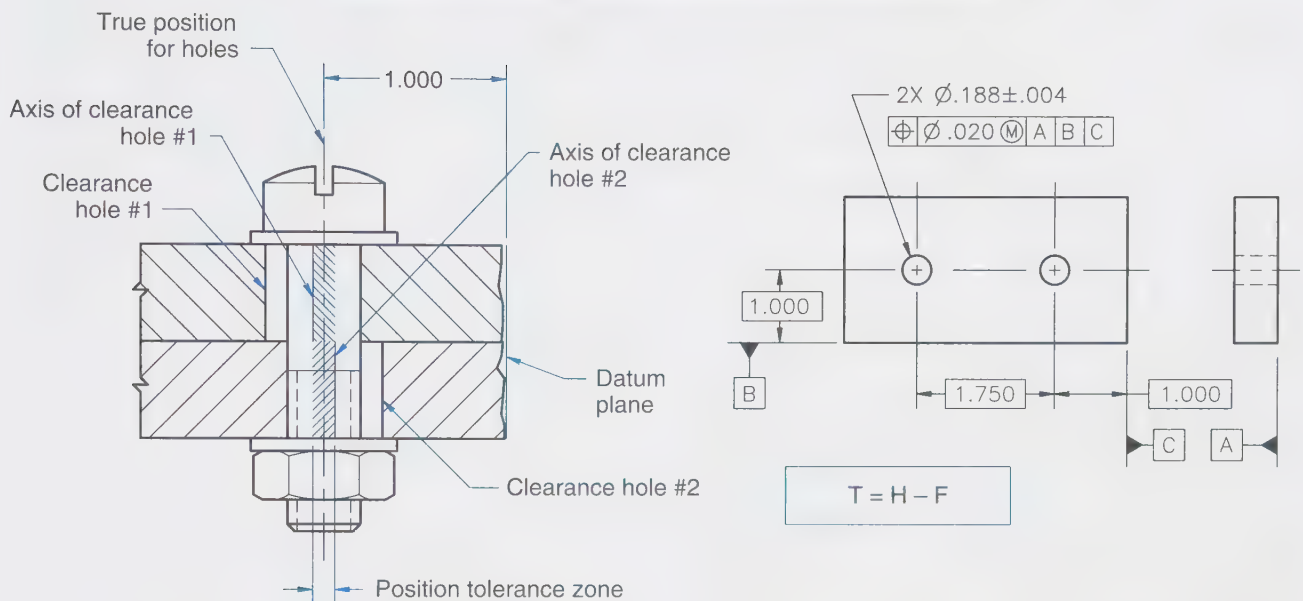
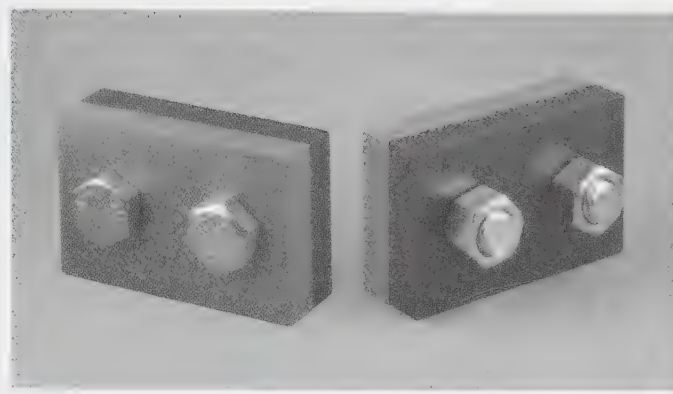
Calculations are completed to determine tolerances that ensure the bolts can be installed through the clearance holes. When clearance holes in two parts are located from common datums, the formula used to calculate the tolerance value is:

$$T = H - F$$

This formula results in a tolerance value that is applied to each of the parts. The letter T is the position tolerance value, H is the MMC of the hole, and F is the MMC of the fastener. **Figure 12-1** shows an enlarged section view of a bolt passing through two clearance holes. The datum features of the two parts are aligned with a common datum plane. A true position axis is located at the basic distance from the datum plane. A tolerance zone for each hole is centered on the true position axis. Each tolerance zone diameter is calculated using the $T = H - F$ formula.

The same size hole is shown in each part. The amount of clearance between each of the holes and the bolt is the same. This clearance is equal to the permitted position tolerance on each of the holes.

The figure shows the axis of each hole moved to opposite extremes within the tolerance zone. This results in opposite sides of the hole making contact with the bolt. Even though the holes are in the worst possible location relative to one another, the bolt is able to pass through the holes. Any other location of the holes, that is within the allowed position tolerance, would be better than the one shown, and would also permit the bolt to pass through. This ensures that the bolt can be installed while the datum features are aligned. It allows the datum features to be aligned during assembly, but it does not force the datum features into alignment.



Goodheart-Willcox Publisher

Figure 12-1. A floating fastener condition has a shaft passing through multiple parts that all contain clearance holes.

Looking at **Figure 12-1**, it can be seen that two worst case parts have clearance on opposite sides of the bolt when the datum features are aligned. If nothing holds the datum features in alignment, it is possible for the two parts to shift in opposite directions from what is shown. The clearances would shift to opposite sides of the bolt and the datum features would be misaligned. The amount of misalignment could be as much as the total clearance in each of the holes, or $2(H - F)$ when the holes are at the MMC size. It is important to know that the datum features can be held in alignment, but also to know that if they are not held in alignment, they can misalign by the amount of the clearance in the holes as explained here.

Produced holes sometimes have a combination of location and orientation variations. See **Figure 12-2**. Provided the axis of a hole falls anywhere within the tolerance zone permitted by

properly completed calculations, the fastener will be able to pass through the clearance holes.

The given example shows two holes produced with orientation variation. Each hole axis falls within the permitted tolerance zone. The fastener does fit through the holes because the amount of clearance between the hole and fastener permits the axis to be at any location or orientation within the tolerance zone.

In the shown example, the holes are at an extreme angle within the tolerance zone, and the fastener is in contact with a point at the top and bottom of each hole. Looking down through the holes, the angle of the holes causes them to appear elliptical. The fastener is making contact across the minor axis of the apparent ellipses. Clearance exists all around the fastener except at the points of contact. The clearance can permit movement of one part relative to the other.

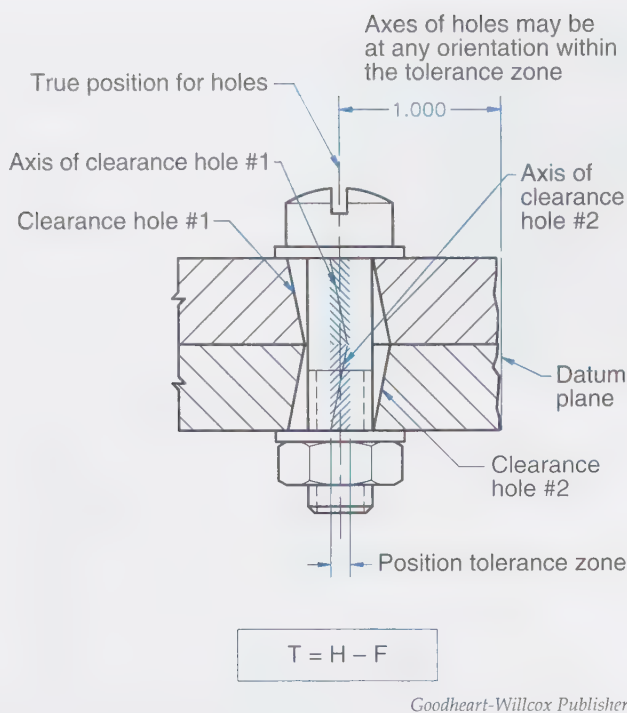


Figure 12-2. The axis of a hole may lie anywhere within the position tolerance specified for that hole.

The ability to achieve alignment of the edges of parts in an assembly is not always required. See **Figure 12-3**. Any time an allowable mismatch between datum features in an assembly exists, tolerances may be calculated for a composite position tolerance. A previous chapter explained that a composite position tolerance creates a relatively large pattern-locating tolerance and a smaller feature-relating tolerance.

The second segment of a composite position tolerance specifies the feature-relating tolerance. For a floating fastener condition, the tolerance value for a feature-relating tolerance (T_F) is usually calculated using the following formula:

$$T_F = H - F$$

This formula works when the primary datum references are to flat surfaces, the holes are perpendicular to the primary datum, and the primary datum reference is repeated in the feature-relating tolerance specification. Calculations completed in this manner ensure that hole locations within a pattern are adequately controlled to permit the fasteners to be installed.

The feature-relating tolerance does not by itself control the allowable mismatch at the edges of the assembled parts. The pattern-locating tolerance directly impacts the allowable mismatch, and the feature-relating tolerance may also result in additional mismatch.

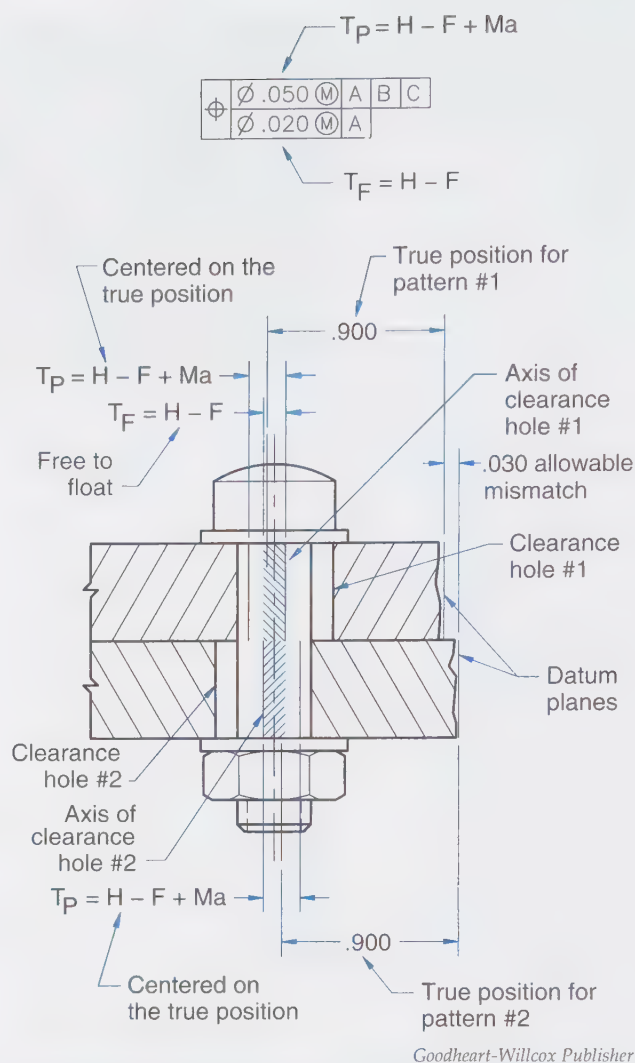


Figure 12-3. A composite position tolerance specification has a pattern-locating tolerance that permits relative movement of the assembled parts and a feature-relating tolerance that ensures the fasteners can pass through the holes.

Calculation of the pattern-locating value (T_P) of the composite position tolerance is completed using the following formula:

$$T_P = H - F + Ma$$

T_P = Tolerance, Pattern-Locating

Ma = Mismatch, Allowable

Pattern-locating tolerance values calculated this way permit the edges of assembled parts to be misaligned and the edges can be held to limit mismatch to the value Ma . It is similar to the single-segment feature control frame ensuring that edges can align. The Ma value ensures the edges can be held to within the Ma value, but clearance in the holes can permit a greater mismatch than the Ma value. Misalignment of any referenced datum

features can be held within the value entered for the Ma value in the formula, provided those features are surfaces and referenced in the same order of precedence in the tolerance specification for both of the parts. **Figure 12-3** shows two parts at one extreme relative to the fastener and the edge misalignment can be held to within the Ma value.

If the two parts are not held in a manner that minimizes the edge misalignment, the clearances in the holes can result in additional misalignment when the parts shift in opposite directions to the one shown. Observing the clearances in **Figure 12-3**, the upper part could possibly be moved to the left and the lower part to the right. The parts can move by a distance equal to the clearances in the holes, and those clearances are equal to the feature-relating tolerance when the holes are at the MMC size. So, the worst case edge misalignment when the holes are at MMC becomes equal to the value $Ma + 2(H - F)$.

One factor that has not been considered in the preceding explanation is the effect of primary datum surface variations on the two parts. If surface variations are large, it is possible that high points on one surface could fit inside low points on the other part. This is unlikely on small machined parts, but more likely to happen to some extent on larger parts. If it happens, the primary planes on the two parts will not coincide. In extreme cases, or when tolerances are very small, this could affect whether the fasteners pass through the holes.

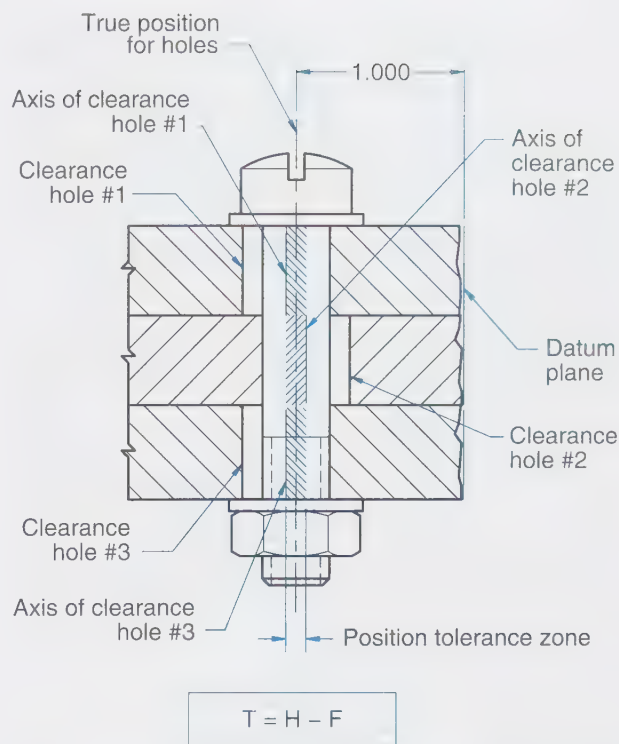
Generally, good quality surfaces are selected as datum features and the error between datum planes on mating parts is so slight that it is unnecessary to adjust position tolerances to accommodate the differences. If, however, the surface variations are large in relation to the surface size, those surface variations need to be considered in the position tolerance calculations.

It is also possible to minimize the surface variations to ensure datum coincidence in an assembly. A form tolerance may be applied to the datum features, or it may be desirable to control the form of the surface by the size tolerance and Rule #1.

Three Stacked Parts

Assemblies may have three parts stacked together. See **Figure 12-4**. When all three parts have clearance holes, a floating fastener condition exists.

The explanation of position tolerance calculations for three stacked parts assumes that parallelism of surfaces on the individual parts is perfect. As the surfaces depart from parallel, the resulting orientation of the holes requires an increase in hole diameter or a reduction in the allowable position



Goodheart-Willcox Publisher

Figure 12-4. If the holes are all equal size, position tolerances for three stacked parts in a floating fastener application may be calculated in the same manner as for two stacked parts.

tolerance. The impact of the parallelism variation is affected by the thickness of the part in relation to the diameter of the hole. If the middle part is a thin gasket, the parallelism variation would most likely be minimal and the effect on clearance through the holes would therefore be negligible in most applications.

Position tolerances for three stacked clearance holes are calculated in the same manner as for two holes. The same calculation methods work under the condition that the holes in all three parts have the same MMC size.

Figure 12-4 shows a position tolerance zone centered on the true position for the three stacked holes. The size of the tolerance zone is determined using the formula $T = H - F$. Each of the holes in the figure is offset to an extreme allowable position. Even with the holes in the shown positions, the fastener passes through the holes.

Provided the same edges on the three parts are identified as datum features and referenced in the same order of precedence, the edges of the parts can be aligned when the fastener is installed. Using different datum references can result in misalignment of the edges; however, the hole patterns will align to permit installation of the fasteners.

As already stated, if the surfaces on the parts are not parallel, the parallelism variation will impact the fit of the fastener in the holes. Any allowable parallelism tolerance should be considered for the effect it will have on the apparent hole size and position tolerances appropriately reduced.

Two Parts, Unevenly Distributed Tolerances

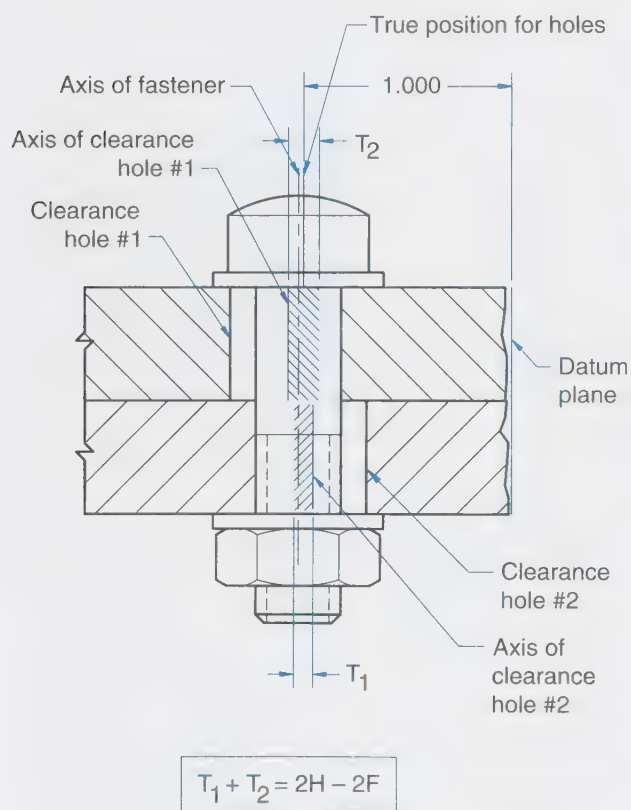
Two parts containing holes of the same size or of different sizes may have the total amount of position tolerance unevenly distributed between the holes. See [Figure 12-5](#). Two parts are shown stacked together in the given figure. The edges of the two parts are aligned. The aligned edges are used as datum features to locate the holes.

Position tolerances for the two clearance holes are calculated using the following formula:

$$T_1 + T_2 = 2H - 2F$$

If the holes are of different sizes, the following formula is used:

$$T_1 + T_2 = H_1 + H_2 - 2F$$



Goodheart-Willcox Publisher

Figure 12-5. The total allowable position tolerance may be distributed unequally when the larger position tolerance is projected or an orientation tolerance is used to refine the larger tolerance.

The resulting tolerances should be carefully applied on the drawing. If the amount of position tolerance applied to one of the holes is greater than would result from the $T = H - F$ formula, it is advisable to apply the position tolerance as a projected tolerance zone. This should be considered because the position and orientation of the fastener is affected by the amount of tolerance in excess of the tolerance permitted by the hole diameter. In effect, the hole acts as a fixed fastener condition (a partially fixed fastener) to the extent of the tolerance that is in excess of the amount permitted by the hole diameter.

The formula in [Figure 12-5](#) can be proved to work on the basis of the formula used when calculating one tolerance value to apply on two holes. The formula $T = H - F$ results in a value for T that is used on two holes. The total tolerance is therefore equal to $T + T$, which is the same as $2T$. To obtain a value of $2T$, the formula can be multiplied by two, giving the formula $2T = 2H - 2F$. Substituting $T_1 + T_2$ for $2T$ in the formula results in the previously given formula of $T_1 + T_2 = 2H - 2F$.

Any portion of the total position tolerance may be applied to one hole. However, if unequally distributed, the larger tolerance value should be applied as a projected tolerance zone.

If unequal distribution of tolerances is used for holes of the same diameter, it should be done with consideration given to the manufacturing requirements. Specification of an extremely small tolerance on one of the two parts will drive the cost upward for that part.

Two Parts, Two Hole Sizes

Designs are sometimes completed using purchased standard, or catalog, parts in combination with newly designed parts. Because purchased parts may have existing holes and position tolerances, it is necessary to design parts with hole sizes and tolerances that will assemble with the purchased parts.

Position tolerances may be calculated when clearance holes in a floating fastener condition are not equal in size. See [Figure 12-6](#). The shown assembly has a large diameter hole in the top plate. The bottom plate has a smaller hole in it. Tolerance zones for the two holes are calculated on the basis of the hole sizes.

One method of calculating position tolerances permits the tolerance for each hole to be calculated separately. This method may only be used if the tolerances on both holes are calculated in the same manner. The two position tolerance values are calculated using the following formulas:

$$T_1 = H_1 - F$$

$$T_2 = H_2 - F$$

$$T_1 = \text{Tolerance, Hole \#1}$$

$$H_1 = \text{MMC, Hole \#1}$$

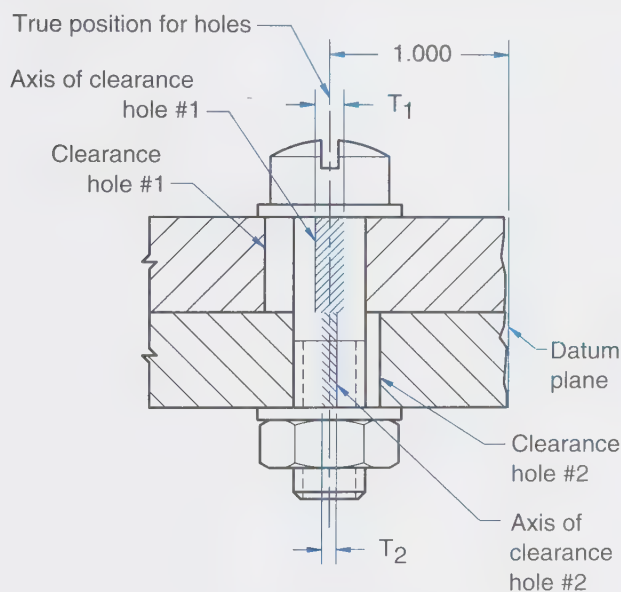
$$T_2 = \text{Tolerance, Hole \#2}$$

$$H_2 = \text{MMC, Hole \#2}$$

Pro Tip

Do not use the previous method unless tolerances on both parts are known to be calculated using this method.

A second method of calculation permits the total allowable tolerance, based on the two hole sizes, to be distributed in any desired manner. Distributing the tolerance also makes it possible to use a small tolerance zone on a large hole while using a larger tolerance zone on a small hole. The



$$T_1 = H_1 - F$$

$$T_2 = H_2 - F$$

Or

$$T_1 + T_2 = H_1 + H_2 - 2F$$

Goodheart-Willcox Publisher

Figure 12-6. The simple formulas shown may be used to calculate tolerances for individual parts provided the same methods are used for all parts in the assembly.

tolerance values may be calculated using the following formula:

$$T_1 + T_2 = H_1 + H_2 - 2F$$

This formula is similar to the one shown for distribution of the available tolerance on two equal size holes. The only difference is that $H_1 + H_2$ has been substituted for $2H$.

When using this method to distribute tolerances, it is advisable to use a projected position tolerance zone for the position tolerance that is larger than the hole size would normally permit.

Fixed Fastener Condition

Fixed fastener condition has been described to some extent in previous chapters. A more in-depth look at the fixed fastener condition makes it possible to see how calculations of tolerances may be completed for simple or complex designs that include fixed fasteners.

A fixed fastener condition exists any time a bolt, pin, shaft, or other feature is fixed in position by a feature on a part. Examples of fixed fastener conditions include a threaded hole into which a fastener is threaded, a press fit for a shaft or dowel pin, and a tapered hole into which a tapered pin is inserted.

More than two parts may be stacked together in a fixed fastener condition. However, all the parts except one must have clearance holes if position tolerances are to be applied to establish interchangeable parts.

The following figures show fixed fastener conditions and calculation methods for determining position tolerances. The calculations shown are accurate. However, the tolerance values are calculated on the basis of using projected tolerance zones. As defined in an earlier chapter, a position tolerance applied to a threaded hole or press fit hole should be applied as a projected tolerance zone. If projected tolerances are not used, calculations must be completed in a manner that compensates for the projection of the fixed location fastener. This results in a tolerance zone that is smaller than when projected tolerance zones are used. It is typically preferable to use projected tolerance zones to avoid reducing tolerance values.

Tolerances for **Figures 12-7** through **12-10** are calculated as though projected tolerance zones are to be specified. Although the true location of the projected tolerance zone lies outside the threaded hole, the figures are simplified and show the tolerance zone within the hole. Showing the projected zone outside the threaded hole would superimpose

the projected zone over the tolerance zone for the clearance hole in the other part. Showing one zone on top of the other would make the figures difficult to understand. It is necessary to show the tolerance zones for both holes in order to provide an understandable explanation. So, these figures are simplified and do not show the tolerance zones as projected. The tolerance zones for the threaded holes in these figures are shown within the threaded holes. This simplification is made possible by illustrating the hole position variations without any perpendicularity variation.

Two Stacked Parts

A single-segment position tolerance specification results in tolerance zones that permit edges of the parts to be aligned. This is true if the same edges of the mating parts are identified as datum features and referenced in the same order of precedence. The edges of the parts are not required to be aligned, but only permitted to be aligned.

The given figure shows a fixed fastener condition for two stacked parts. See **Figure 12-7**. A basic dimension is given to locate the true positions for

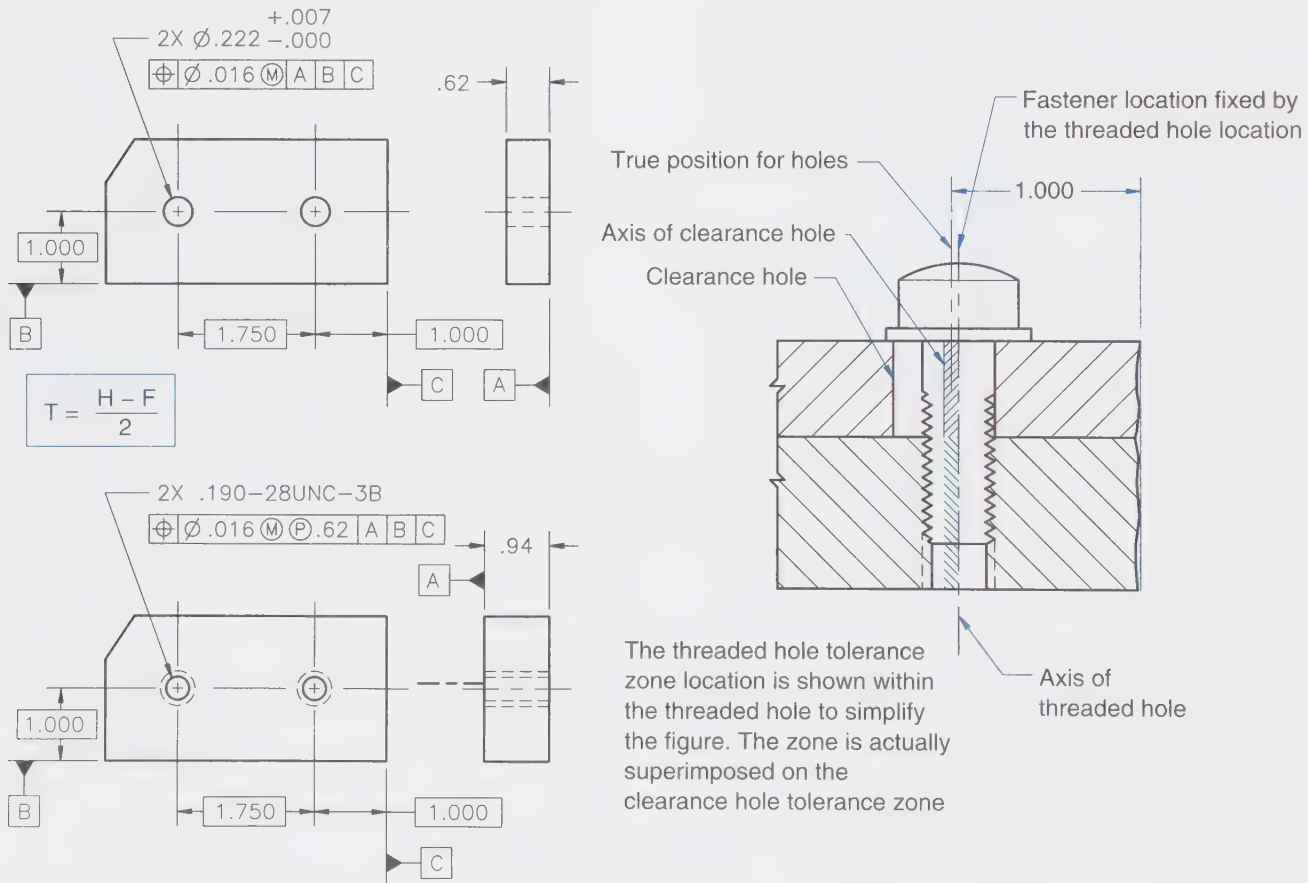
the two holes. A tolerance zone for each hole is centered on the true position. Calculation of the diameter for the two tolerance zones is completed using the following formula:

$$T = (H - F)/2$$

Tolerance value T is applied to each of the holes. The previous formula is only used to determine the tolerance value if each part is to be assigned the same amount of tolerance. It may only be used if a projected tolerance zone is specified for the threaded hole.

Tolerance values for a particular clearance hole diameter used in a fixed fastener condition are only half the allowable tolerance value for the same size clearance hole used in a floating fastener condition. This is true because the position variation of the fixed fastener feature utilizes part of the clearance between the fastener and the clearance hole.

Figure 12-7 shows the threaded hole positioned at one extreme of the specified tolerance zone. The threaded hole location determines the fastener location. Movement of the fastener off



Goodheart-Willcox Publisher

Figure 12-7. Fixed fastener calculations using the shown formula require the utilization of a projected tolerance zone on the fixed fastener hole. The tolerance zone for the threaded hole is not shown projected to simplify the figure.

the true position utilizes some of the clearance between the clearance hole and the fastener.

The given figure shows the clearance hole located at the opposite extreme of the tolerance zone relative to the fastener location. Assembly of the fastener through the clearance hole is possible if the fastener and clearance hole are displaced to opposite extremes of their allowable tolerance.

Figure 12-7 shows equal tolerance zones on both the threaded hole and the clearance hole. To simplify the illustration, the tolerance zone for the threaded hole is shown within the threaded part. (In reality, the tolerance zone should be projected for the threaded hole, and therefore would lie outside the threaded part.) Because the tolerance zone for the threaded hole is shown within the threaded part, the orientation of the fastener is maintained perpendicular to the interface surface. This prevents any fixed fastener effects that would result from orientation variation.

The datum features in the given example may be held so they are aligned when the fastener is installed. If the datum features are not held aligned, the assembled parts may shift, allowing an edge misalignment that could be equal to the clearance between the fastener and the clearance hole.

Requirements for a design often permit the edges of parts to be misaligned to some extent. See **Figure 12-8**. Composite position tolerances may be used to increase tolerance zones when edge misalignment is permitted.

Feature-relating tolerance (T_F) values are calculated using the same formula that is used to obtain edge alignment, assuming the datum features are surfaces and the primary datum feature reference is repeated for the feature-relating tolerance. Pattern-locating tolerance (T_P) values are calculated using the following formula:

$$T_P = (H - F)/2 + Ma$$

Values calculated must be applied to the holes in each of the two parts. With a pattern-locating tolerance applied, it is known that the datum features can be held in alignment within the distance Ma . However, the parts may shift if not held in place, and the misalignment may be as much as $Ma + (H - F)/2$ when the clearance hole is at MMC.

Three Stacked Parts

Tolerances for a fixed fastener application with three parts stacked together may be calculated much in the same manner as for two stacked parts

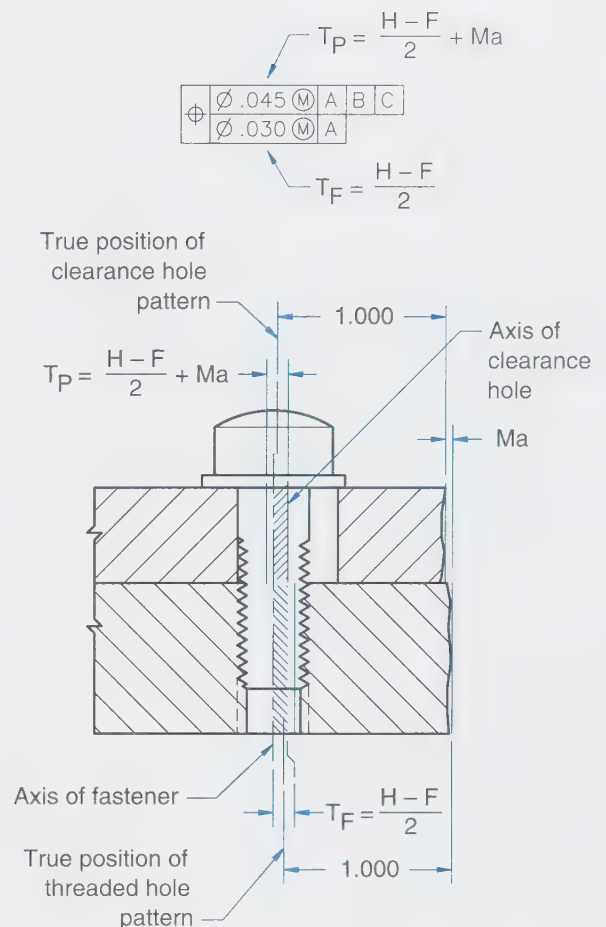


Figure 12-8. Composite position tolerances may be used for fixed fastener conditions to allow relative part movement as was explained for floating fastener applications.

if the two clearance holes have the same MMC size. See **Figure 12-9**. The edges of the parts in the given figure are aligned. Calculating tolerances using the shown formula results in a tolerance value that must be specified in a single-segment feature control frame if alignment of the edges is to be maintained. The tolerance on the threaded hole must be specified to project a distance equal to the combined thickness of the parts containing the clearance holes. If the faces of the parts are subject to parallelism variation, that variation has the potential to reduce the allowable position tolerance.

The threaded hole in the shown figure is positioned at one extreme of the allowable tolerance zone. Both of the clearance holes are shown at the opposite extreme. This is a worst-case condition because any other position of either hole would result in clearance all around the fastener.

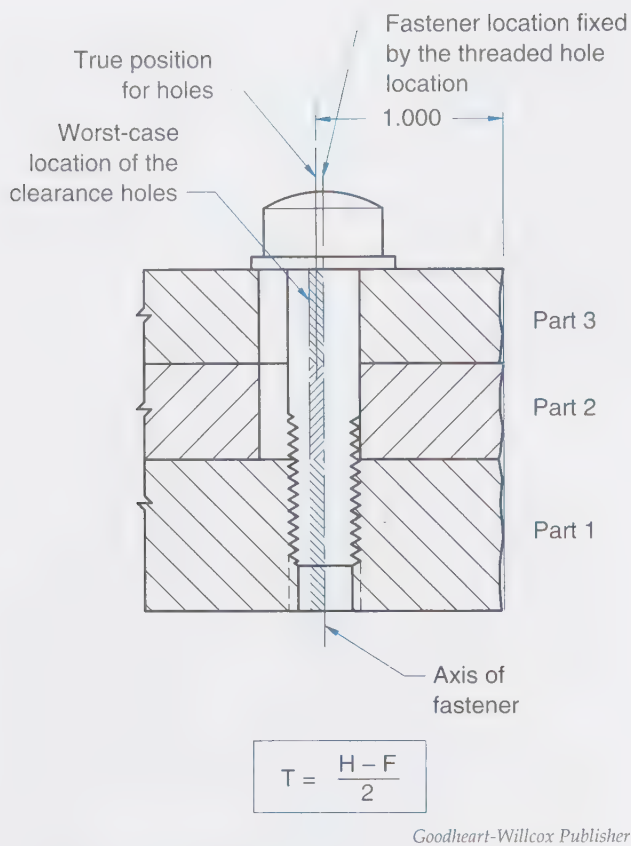


Figure 12-9. Position tolerances for multiple parts in fixed fastener applications are calculated for pairs of parts, always using the fixed fastener feature as one of the parts in each pair.

Two Parts, Unevenly Distributed Tolerances

Holes in a fixed fastener condition may have the total allowable position tolerance unevenly distributed between the holes. See **Figure 12-10**. Two parts are shown stacked together in the given figure. The edges of the two parts are aligned. The aligned edges are used as datums to locate the holes.

Position tolerances for the two holes are calculated using the following formula:

$$T_1 + T_2 = H - F$$

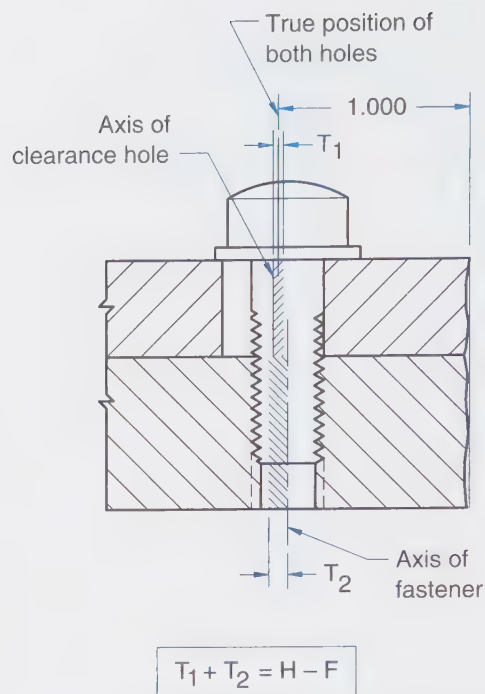
This formula can be proved to work on the basis of the formula used when calculating a single tolerance value to apply on two holes. The formula $T = (H - F)/2$ results in a value for T that is used on both holes. Multiplying the formula by 2 results in:

$$2[T = (H - F)/2]$$

$$2T = H - F$$

Because $2T$ is equal to $T + T$, by substitution the formula may be rewritten as:

$$T_1 + T_2 = H - F$$



Goodheart-Willcox Publisher

Figure 12-10. The total allowable position tolerance may be distributed between the features in a fixed fastener application.

This formula permits any portion of the clearance between a hole and fastener to be applied to the fastener location and to the clearance hole location with the condition that the position tolerance on the fixed fastener is applied as a projected tolerance. The position tolerances on the clearance hole and the fixed fastener hole should total no more than the available clearance between the fastener and clearance hole.

If unequal distribution of tolerances is used, it should be done with consideration given to the manufacturing capabilities. Generally, threaded holes are permitted more of the total position tolerance than the clearance hole. One reason for this is that threaded holes include more manufacturing steps and are not as easy to locate as clearance holes. The other reason is that clearance holes normally have relatively large size tolerances, and the MMC modifier may be utilized to provide bonus tolerances for the location of the clearance hole.

Three Parts, Unevenly Distributed Tolerances

Unevenly distributed tolerances may also be used for three stacked parts when a fixed fastener condition exists. Refer back to **Figure 12-9**. Calculations are completed for two parts at a time. Tolerances for Part 1 and Part 3 in the given figure may be calculated using the following formula:

$$T_1 + T_3 = H_3 - F$$

After tolerances for the first pair of parts (Parts 1 and 3) are calculated, a position tolerance for the remaining part (Part 2) must be calculated. The tolerance calculated for Part 1, in the previous step, must be utilized when calculating the tolerance for Part 2. This is necessary because Part 1 has the fixed fastener condition. The formula used is:

$$T_1 + T_2 = H_2 - F$$

Application of the position tolerance on Part 1 should show a requirement for a projected tolerance zone that extends a distance at least equal to the thickness of the two attached parts.

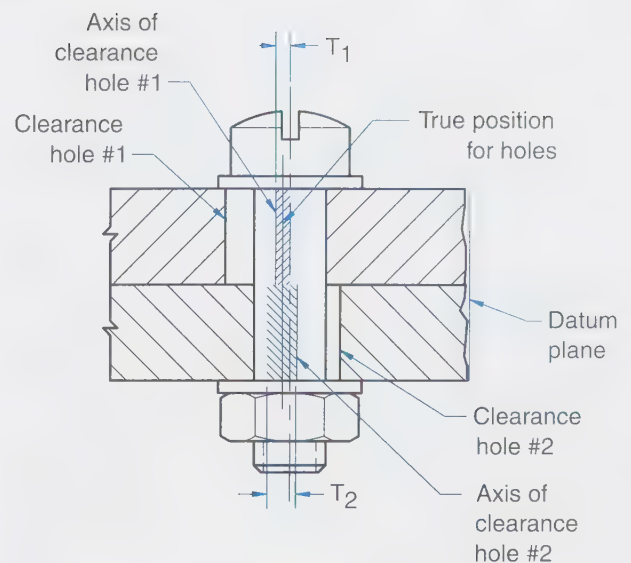
Partially Fixed Fastener Condition

A partially fixed fastener condition exists whenever a hole is assigned a position tolerance that is greater than the difference between the hole MMC and the fastener MMC. Once the positional variation on the hole exceeds the clearance between the hole and the fastener, the fastener location or orientation will be affected by the hole location or orientation.

A potential for a partially fixed fastener condition exists any time the positional tolerances are calculated based on the total clearance in the two holes and that tolerance is distributed between the parts. See **Figure 12-11**. The given figure shows two parts with clearance holes of different sizes and a fastener passing through the holes. Hole #1 in Part 1 has a large clearance relative to the fastener. Hole #2 in Part 2 has a smaller amount of clearance. The total clearance that exists in the two parts is used to calculate the total position tolerance that may be distributed between the parts using the formula in the given figure.

The shown formula only results in a floating fastener condition if the tolerances are based on the clearance between each hole and the fastener that passes through it. Any other distribution will result in one of the two parts partially fixing the fastener location. Whichever hole gets more positional tolerance than its clearance permits will be the hole that affects the fastener location. If T_1 is greater than H_1 minus F , hole #1 will cause a partially fixed fastener. If T_2 is greater than H_2 minus F , hole #2 will cause a partially fixed fastener.

In **Figure 12-11**, hole #2 is provided a large position tolerance relative to the amount of clearance it provides. The hole is shown at the maximum allowed position error relative to the true position. The result is that hole #2 pulls the fastener off its true position.



$$T_1 + T_2 = H_1 + H_2 - 2F$$

Goodheart-Willcox Publisher

Figure 12-11. A partially fixed fastener condition exists when a position tolerance is larger than the clearance between a hole and fastener. The larger position tolerance can result in pulling the fastener off true position.

Hole #1 has a smaller position tolerance applied to it than the clearance would permit. This is required because hole #2 was assigned more tolerance than its clearance would permit.

When it is desired to use a larger position tolerance than is permitted by the hole size, either of two actions will accommodate the partially fixed fastener condition. It is possible to apply a projected tolerance zone just as was previously explained for a fixed fastener application. It is also possible to apply an orientation tolerance (using a projected tolerance zone) that is equal to the specified position tolerance (T) minus the clearance in the hole. So, the orientation tolerance should be equal to $T - (H - F)$. This will prevent the orientation of the fastener from interfering with the mating clearance hole.

The need to use a partially fixed fastener position tolerance may be caused by a situation where one part is purchased and the requirements for hole size and position tolerance are already established. The design of a mating part must accommodate the dimensional requirements for the existing part. Another situation could exist when designing parts that are significantly different in complexity, or when parts are designed for significantly different manufacturing processes.

It is preferable to avoid partially fixed fastener conditions. To avoid a partially fixed fastener situation, the position tolerance and hole size may be calculated for each part using the following formulas.

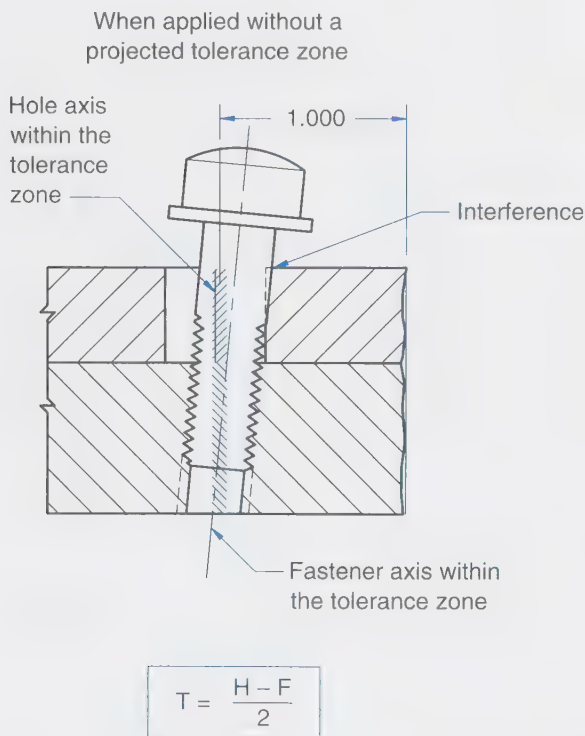
$$T_1 = H_1 - F$$

$$T_2 = H_2 - F$$

Projected Tolerance Zone

Position tolerances applied to features that result in a fixed fastener condition usually require a projected tolerance zone. Failure to specify a projected tolerance zone results in a tolerance zone that resides within the boundary of the feature to which the tolerance is applied. Failure to use a projected tolerance zone can result in an interference condition when the $T = (H - F)/2$ formula is used.

See **Figure 12-12**. The given figure shows the results of omitting the requirement for a projected tolerance zone. The position tolerances applied to the threaded hole and the mating clearance hole were calculated using the $T = (H - F)/2$ formula, and applied without the requirement for a projected zone. The tolerance zone is therefore contained within the threaded hole.



Goodheart-Willcox Publisher

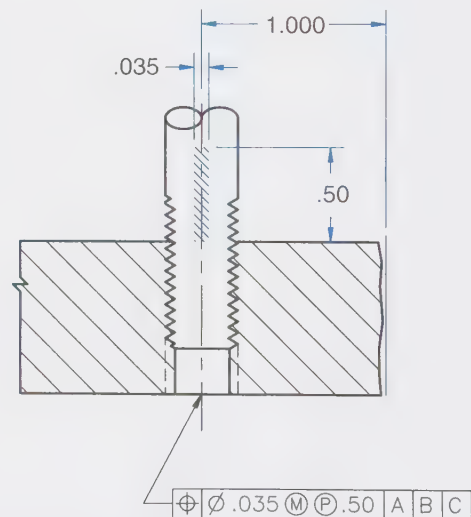
Figure 12-12. Failure to use a projected tolerance specification for a fixed fastener application can result in an interference condition.

The axis of a hole is permitted to lie anywhere within the specified tolerance zone. As shown in the figure, the axis of the hole may include orientation variation. Because a bolt threaded into a hole will align on the axis of the threads in the hole, the bolt will have the same orientation variation that exists in the hole.

A clearance hole located at an extreme within the permitted tolerance zone is shown in the figure. When the hole is in an extreme location, it is possible for a fastener installed in the threaded hole to interfere with the edge of the clearance hole. If this occurs, the parts will not assemble.

Application of a projected tolerance zone on a threaded hole does, in effect, control the location of a fastener that is threaded into the hole. However, it controls the location of the fastener on the outside of the threaded hole. See **Figure 12-13**. As explained in a previous chapter, a **projected tolerance zone** specification establishes a tolerance zone that resides outside the controlled feature. In the given example, the threaded hole includes a .035" diameter tolerance zone that is to apply for a distance of .50" outside the part.

The threaded hole must be located so that its axis lies somewhere within the tolerance zone that is projected outside the part and centered on the true position of the hole. The tolerance zone in the given figure is .035" diameter and extends from the surface to a distance .50" above the part. Controlling the location of the hole in this manner ensures that the fastener threaded into the hole has a known location outside the controlled part.



Goodheart-Willcox Publisher

Figure 12-13. A projected tolerance specification controls the axis of the controlled feature only in the projected zone.

Tolerance specifications on fixed fastener features should include projected tolerance zone requirements. See **Figure 12-14**. Projected tolerance zone requirements result in superimposed position tolerance zones for the clearance hole and the fixed fastener feature. If the axis of the threaded hole falls within the projected tolerance zone, then the axis of the fastener will also fall within this zone. This ensures the fastener will pass through the clearance hole.

The projected tolerance zone may be specified in the feature control frame when the direction of projection is obvious. The projected tolerance zone symbol is the letter P inside a circle. It follows any material condition modifier. The projection distance follows the symbol.

When the projection direction is not obvious in an orthographic view, a chain line is applied adjacent to the controlled feature to indicate direction. It is also permissible to dimension the distance on the chain line and omit the distance from the feature control frame. A chain line is not used on CAD models. For a model, the projected zone

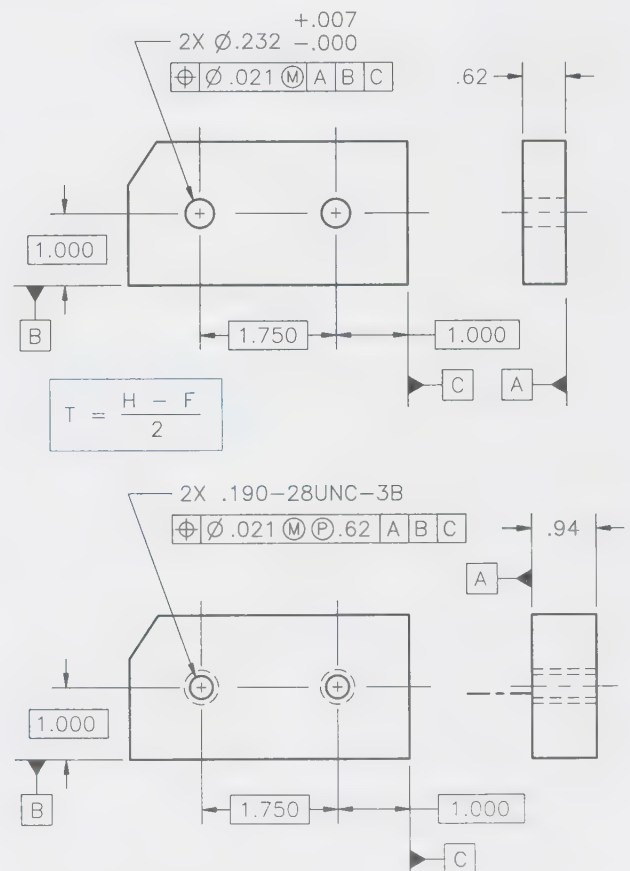
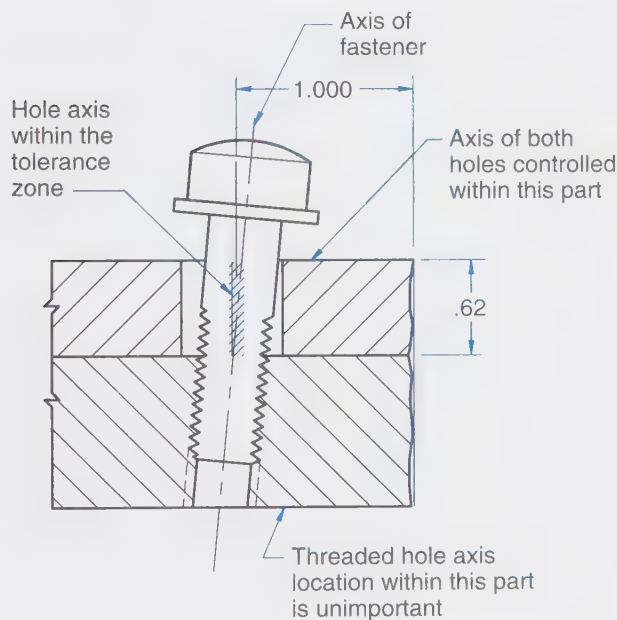
extends from the surface where the leader line terminates on the hole.

Multiple Parts in Assembly

A complete assembly is often made up of parts that stack together in more than one plane. See **Figure 12-15**. The shown assembly is a six-sided enclosure. Four side panels bolt together to form a complete set of side walls. A top and bottom cover are bolted to the side panels.

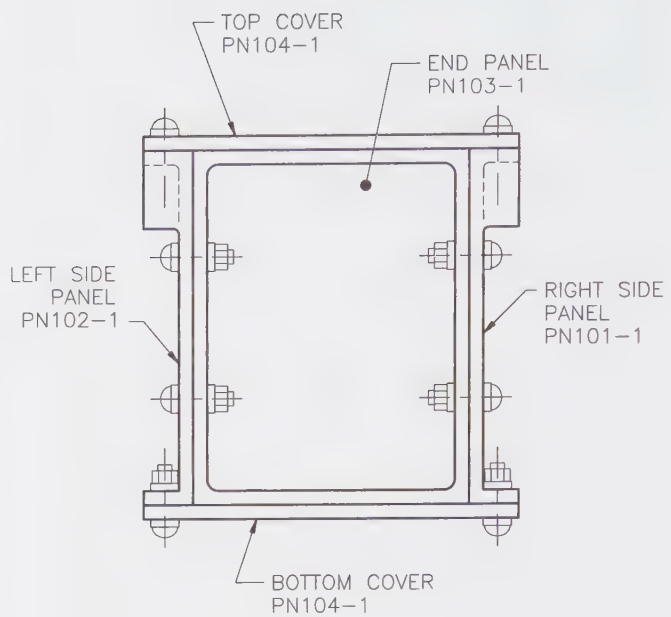
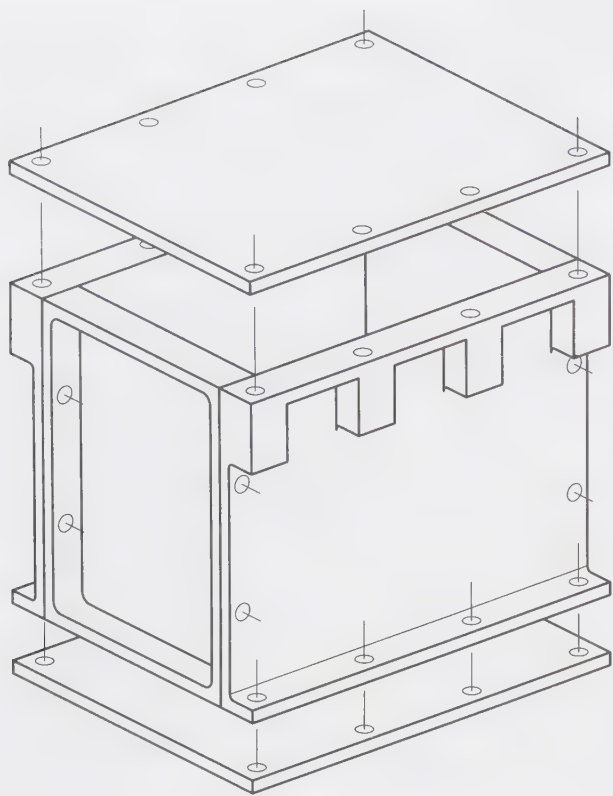
Holes in the covers must align with those in the side panels before bolts can be installed to hold the covers in place. Alignment is only possible if the accumulation of tolerances between the side walls and covers is controlled. The holes must be sized according to the amount of tolerance accumulation.

Details of how to calculate and apply tolerances to the shown figure are described in this chapter. Completion of calculations for any assembly requires the assembly be thoroughly understood before starting calculations.



Goodheart-Willcox Publisher

Figure 12-14. A projected tolerance zone ensures the fastener axis is controlled where it extends through the part containing the clearance hole.



Goodheart-Willcox Publisher

Figure 12-15. How tolerances in an assembly accumulate must be considered when calculating tolerances and applying them on the drawings.

The shown assembly has two side panels with threaded holes for attachment of the covers. The left side panel is identified as part number PN102-1. The right side panel is identified as part number PN101-1. Two end panels identified as PN103-1 are placed between the two side panels.

Each of the side panels must include a position tolerance on the threaded cover attachment holes. The top and bottom covers must also include a position tolerance on the clearance holes. Tolerances on the side panel hole locations and on the cover clearance holes affect assembly of the covers to the side walls. Also, there is a tolerance on the length of the end panels, and this tolerance affects the distance between side panels. The tolerance on the distance between side panels affects the relative distance between cover mounting holes on the assembled side panels. Parallelism of the sides on the end panels are within the length tolerance and for this application will not be a significant factor. Perpendicularity of surfaces is assumed to have no variation for purposes of this example. Perpendicularity variation would result in a box shape that is distorted. If needed, the 90° corners of the box may be maintained with tooling to prevent the ends of

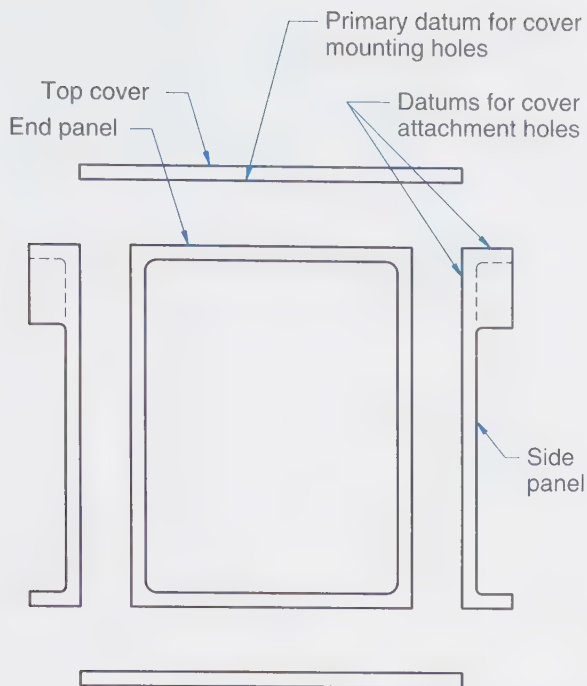
the panels from impacting the fit of the cover. It may also be necessary to install the cover before the side panel fasteners are tightened.

Selection of Datums

Total tolerance accumulation in an assembly such as the one shown in **Figure 12-15** must be minimized through selection of the best design concept and by following well-planned assembly processes to keep clearance hole sizes at a reasonable diameter and to permit assembly of the parts. One contributor to minimizing tolerance accumulation is the proper selection of datum features.

Datum feature selection should always include consideration of the functional features in the design and should also take into consideration the fabrication and assembly processes. See **Figure 12-16**. The figure shows an exploded view of parts in an assembly. For an assembly such as this, identification of functional features is relatively simple.

The top cover only has one surface that comes in contact with the side walls and all the holes are perpendicular to this surface. This surface should



Goodheart-Willcox Publisher

Figure 12-16. Datums are normally selected on the basis of how the parts in an assembly fit together.

therefore be identified as a primary datum feature for specification of position tolerances applied to the holes on this part. Edges of the cover may be used as the secondary and tertiary datum features. If it was desirable to control symmetry of the hole pattern relative to the edges of the cover, the width and length could be identified as datum features of size.

The right side panel has one surface that is in contact with the top cover. It has another surface that makes contact with the end panel. These two surfaces should be identified as datum features for locating the cover attachment holes. One of these surfaces will be selected as a primary datum feature, and the other will be used as a secondary datum feature. An end of the side panel will be used as a tertiary datum feature.

Datum feature selection on the end panels is not necessary for cover attachment holes because the end panels do not include cover attachment holes. It is necessary to identify datum features on the end panels for other purposes, such as specification of orientation tolerances and side panel attachment hole position tolerances, but that is not part of the explanation for establishing datums and tolerances to attach the cover. The methods explained for cover attachment calculations may be followed to determine side panel datums and tolerances.

Calculation of Tolerances for an Assembly

Selection of the proper datum features makes it possible to keep accumulation of tolerances to a minimum. By showing how to calculate tolerances for the given assembly, the following paragraphs will also show that the proper selection of datum features as previously explained does minimize tolerance accumulation.

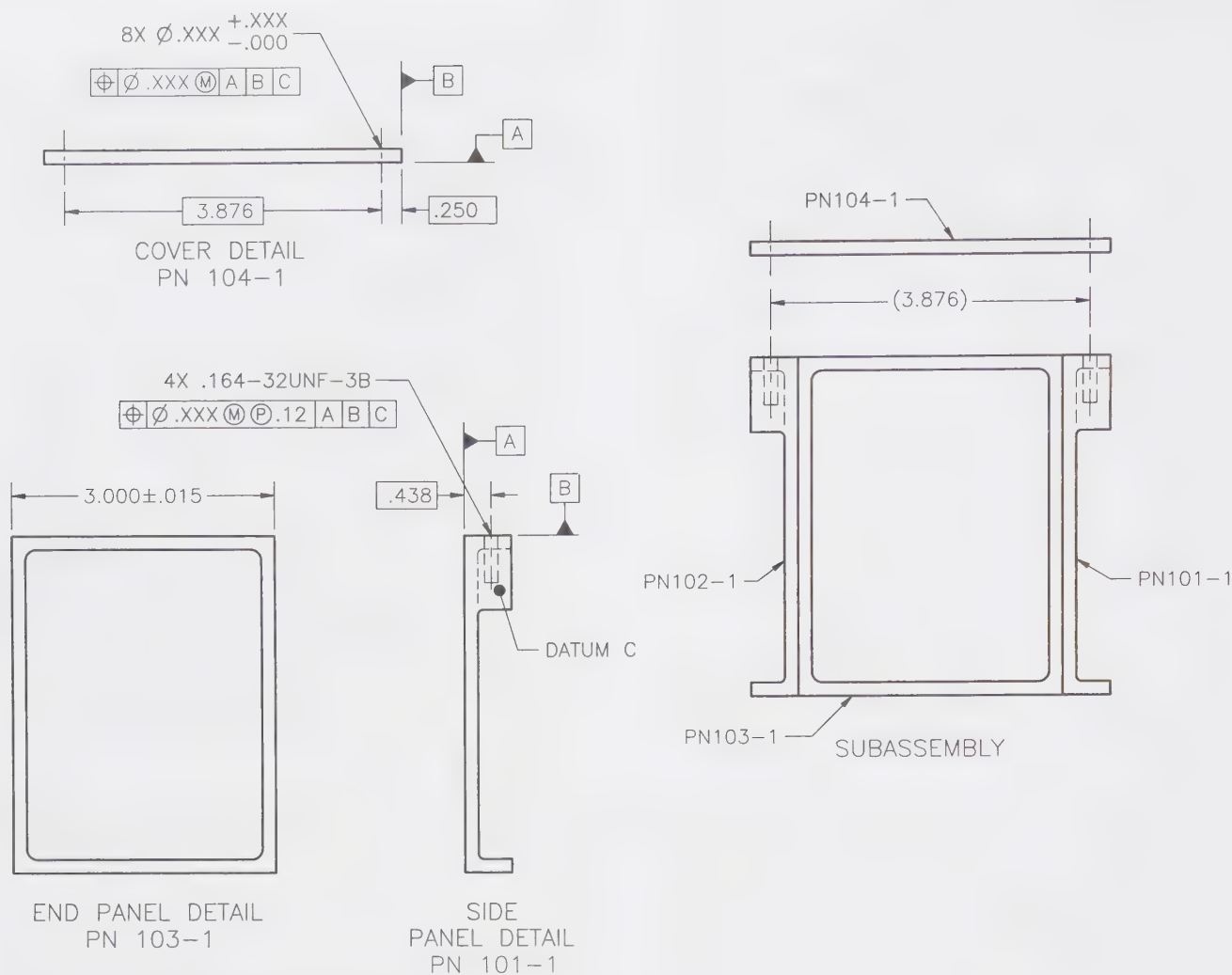
Figure 12-17 shows the top cover located above the side walls subassembly. Centerlines for the cover attachment holes are shown. On the top cover, there is a basic dimension that locates the distance between the clearance holes; that dimension is 3.876".

The side walls subassembly shows the centerlines for the threaded cover attachment holes. These holes are located by dimensions on three parts that total 3.876", which is the same distance between the clearance holes in the cover. Because three parts are involved, tolerances on those three parts must be considered to determine the total variation on the 3.876" distance on the side walls subassembly.

Locating surfaces are normally used as datum features when dimensioning parts. A detail view of one side panel is shown in the figure to show the selected datum features on the side panels. Datum feature symbols are shown. These datum features are referenced in the position tolerance specifications for the cover attachment holes. Each side panel has a basic dimension of .438" that locates the threaded holes from the surface that contacts the end panel, and it is identified as datum A.

The figure shows that the end panel has a size dimension of $3.000" \pm 0.015"$ applied, and the variation on this dimension will affect the positions of holes in the side panels. Each of the side panels includes a .438" hole location dimension. The total of the dimensions applied to the side panels and the end panel equals the 3.876" distance between attachment holes on the cover.

An accumulation of tolerances does take place because of the tolerances applied to the three parts assembled to establish the 3.876" dimension. All tolerances must be carefully considered to determine their effects. There is a position tolerance on each of the side panels, and that tolerance permits location variation relative to the side panel datums. There is also an end panel size tolerance that affects the distance between the side panels, which affects the distance between holes. To simplify this problem, the form and orientation



Only one view of each detail is shown to illustrate the accumulation of tolerances in one direction

Goodheart-Willcox Publisher

Figure 12-17. Tolerances on the side panels, end panels, and top cover affect the size clearance holes that must be produced in the cover.

tolerances on the end panel are initially ignored and therefore eliminated from the explanation of the tolerance accumulation. In fact, the end panels will initially be eliminated from the assembly to simplify the explanation for this assembly.

Position tolerance effects on the two side panels are considered first. See [Figure 12-18](#). Elimination of the end panels from the assembly permits the datum features on the side panels to be brought together. For practical purposes, this creates a common datum reference frame for the two parts. However, if the two datum features had large surface variations, the datums might not perfectly coincide when the datum features make contact.

In theory, the basic dimensions locating the holes on the two side panels extend from a common datum reference frame. Because the position

tolerances on the two groups of holes are, in effect, referenced to a common datum reference frame, the holes may be treated as a single pattern for tolerance calculation purposes. This means there is no accumulation of tolerances on the two side panels.

Selection of datum features to completely eliminate a tolerance accumulation between two parts is not always possible, but it is always possible to minimize tolerance accumulation. Proper selection of datum features will reduce tolerance accumulation.

When the end panel is inserted between the two side panels, an accumulation of tolerance will take place because of the $\pm .015$ " size tolerance on the end panel. See [Figure 12-19](#). Calculation of position tolerances and clearance hole sizes for the



Figure 1 is a dimensional drawing of two components, PN102-1 and PN101-1. The drawing includes the following details:

- Top View (PN102-1):**
 - Overall width: 3.876
 - Feature **H**: $8X \varnothing .XXX \begin{smallmatrix} +.XXX \\ -.000 \end{smallmatrix}$
 - Feature **F**: $.164-32UNF-3B$
 - Feature **T1**: $\varnothing .012 \text{ (M) A B C}$
 - Feature **T2**: $\varnothing .018 \text{ (M) P .12 A B C}$
 - Feature **T3**: $3.000 \pm .015$
 - Feature **A**: $.438$
 - Feature **B**: $.438$
- Side View (PN101-1):**
 - Overall height: $3.000 \pm .015$
 - Feature **A**: $.438$
 - Feature **B**: $.438$
- Other Dimensions and Features:**
 - Feature **C**: $.12$
 - Feature **P**: $.12$
 - Feature **M**: M
 - Feature **A**: A
 - Feature **B**: B
 - Feature **C**: C

$$T_1 + T_2 + T_3 = H - F$$

$$.012 + .018 + .015 = H - .164$$

$$.012 + .018 + .015 + .164 = H$$

.209 Hole (MMC)

Goodheart-Willcox Publisher

assembly must take into consideration the size tolerance on the end panel.

The formulas previously defined for position tolerance calculation may be used for an assembly that has an accumulation of tolerances if the formula is expanded. The formula for a fixed fastener application is:

$$\text{Total tolerance } (T_1 + T_2) = H - F$$

For a floating fastener condition, the formula is:

$$\text{Total tolerance } (T_1 + T_2) = 2H - 2F$$

To expand the formula, all the tolerance values impacting the total position tolerance value are put in the formula.

Figure 12-19 includes three tolerances that impact the assembly of the cover on the side walls. One tolerance is the position tolerance on the threaded holes—both side walls have the same position tolerance and act as one pattern in this example. Another tolerance is the position tolerance on the clearance holes on the cover. The third tolerance is the size tolerance on the end panel. Because there are three contributors to the total tolerance and the application is a fixed fastener condition, the formula for this assembly is:

$$T_1 + T_2 + T_3 = H - F$$

Either the tolerances or the hole size may be assumed, leaving the other value to be calculated. The fastener size of .164" is a fixed value based on standard fastener size. Calculations for the given assembly are completed in this example by assigning position tolerances to apply on the holes. So, with the fastener size known and the position tolerances assigned, only the hole size remains to be determined. The given figure shows how the calculations are made.

Two of the tolerances applied in the calculation are the position tolerances of .012" and .018" diameter. These are values shown in the feature control frames. The third tolerance used in the calculation is the .015" bilateral size tolerance. The value of .015" is used because the size tolerance can only cause a worst-case difference in the nominal location of the holes of .015". Although the $\pm .015$ " tolerance is a total size tolerance of .030", it cannot cause a .030" movement of the holes relative to nominal. Should an end panel be produced at $+.015$ ", the holes on both side panels would in effect see $+.0075$ " movement, requiring .015" diameter clearance in the cover holes. Solving the equation results in a .209" MMC diameter for the clearance holes in the cover.

Zero Position Tolerance at MMC

Previously shown position tolerance calculations have been based on the use of clearance holes that have an MMC larger than the fastener MMC. This is a common practice, and is far better than calculating plus and minus tolerances for hole locations. The advantages of position tolerances were explained in Chapters 8 and 9. Position tolerances, when properly calculated and applied, can provide increased tolerance zones and thereby improve the freedom in the manufacturing of parts.

A further increase in manufacturing freedom may be obtained through the application of a concept known as *zero tolerancing at MMC*. It may not be intuitively obvious, but proper application of *zero tolerancing at MMC* increases manufacturing freedom by allowing the total permitted variation to be applied where needed during manufacturing. Care must be taken not to misapply the concept or it will reduce the tolerances to zero.

The zero position tolerance at MMC concept should only be applied when clearance conditions exist. It may only be used for tolerances that include the MMC or LMC modifier, and there must be a sufficiently large size tolerance on the feature(s) where the zero position tolerance at MMC is applied.

The application of position tolerances as already defined in previous segments of this text is significantly better than using plus and minus location tolerances. In fact, the application of plus and minus tolerances on the locations of holes has no standardized meaning. Position tolerances are far better because a properly calculated position tolerance provides more permissible variation

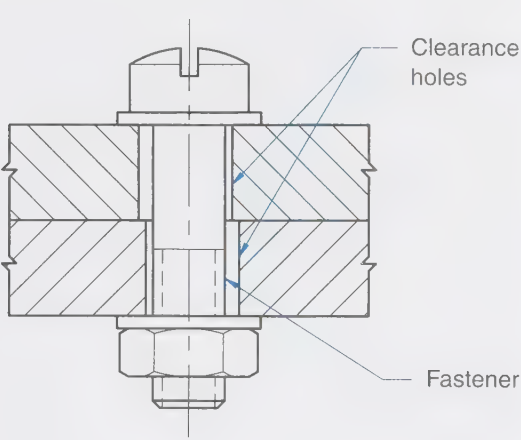
than can be allowed by a plus and minus tolerance that creates a square zone. Perhaps more importantly, with position tolerances there is a known framework of true positions for the holes and such a framework is not achievable with plus and minus tolerances.

The application of zero position tolerances at MMC may result in even greater permissible freedom in how the allowable variation is allocated, and it can be done without any impact on the functional quality of the produced parts. It will ensure that all usable parts are acceptable.

Using only the information defined to this point will result in tolerances that ensure that parts are functionally good. However, those practices may also result in tolerance specifications that require rejection or rework of some functionally acceptable parts. Some produced parts may have conditions where holes are smaller than the minimum size limit, but yet function as needed for the design application. Although the parts will assemble, the small holes would not meet drawing requirements and require rejection or rework.

Rather than use tolerance specifications that may result in the rejection of parts that are functionally acceptable, it would be better to use tolerance specifications that accept all good parts. The following examples show a comparison of methods for applying position tolerances. The zero position tolerance at MMC method results in a specification that accepts all usable parts.

See **Figure 12-20**. A floating fastener application is shown. A position tolerance of .032" diameter at MMC is calculated for the .282" diameter MMC holes. A size tolerance of +.006" is applied to the holes. The fastener diameter is .250".



$$\begin{matrix} \text{+}.006 \\ \text{---}.000 \end{matrix}$$
$$\varnothing .282$$

\varnothing	$\varnothing .032$	M	A	B	C
---------------	--------------------	------------	---	---	---

	Produced hole diameter	Allowable position tolerance
MMC	.282	.032
	.283	.033
	.284	.034
	.285	.035
	.286	.036
LMC	.287	.037
	.288	.038

Goodheart-Willcox Publisher

Figure 12-20. A hole that will correctly function may be rejected if it is produced under the minimum size limit for the hole when tolerances are overly restrictive.

A table in the figure shows the acceptable position tolerance for each allowable hole diameter from the .282" minimum through the .288" diameter. Any hole produced to the requirements of the table is acceptable.

Any hole produced outside the size range or the allowable position tolerance shown in the table is not acceptable. As an example, a .280" diameter hole (too small) produced at a position tolerance of .015" diameter is not acceptable according to the hole specification. The hole does not meet the minimum size requirement of .282". According to the drawing requirement, the part must be rejected. To salvage the part, it is possible to rework the diameter of the hole to make it meet the size requirement, but that means an additional machining operation and another inspection cycle. These extra efforts cost time and money.

If the drawing requirements for the hole are ignored, it can be determined that the .280" diameter hole with a position variation of only .015" diameter permits the fastener to pass through. In fact, a .250" diameter fastener will fit through a .280" diameter hole if it has a position variation of as much as .030". It is reasonable to ask why a drawing specification would be created in such a way that it requires rejection of a part that will work.

The zero tolerance at MMC concept permits specification of hole sizes and position tolerances that allow the full range of functionally acceptable parts to be produced. See Figure 12-21. This figure is the same as the previous one, except the hole specification and table were revised. A .000" diameter position tolerance zone at MMC is specified, and

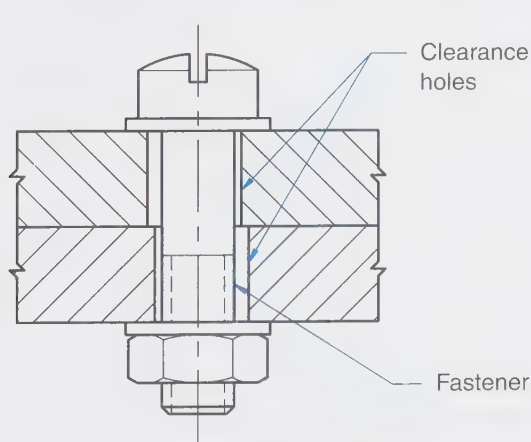
the hole has an MMC equal to the fastener diameter. The hole size has a size tolerance of +.038".

The hole size specification and zero tolerance at MMC application permit the same maximum diameter hole (.288") and the same position tolerance (.038") on the maximum size hole as were permitted in the previous figure. In fact, the entire range of hole sizes and the associated position tolerances shown in the previous figure are permitted by the hole specification in this figure. Additionally, all hole diameters between .250" and .282" diameter along with their associated position tolerances were made acceptable by specifying the zero position tolerance at MMC in the manner shown here.

A significant amount of manufacturing freedom is provided through the proper use of zero tolerancing at MMC. Any functionally acceptable hole may be accepted without rework of the parts. No functionally acceptable parts must be rejected because all functionally acceptable parts meet the hole specification.

There are a couple of precautions that should be taken when specifying zero position tolerances at MMC. The first is to make sure that all people working to the drawing understand the MMC concept. If the MMC concept is not understood, people may think the zero tolerance specification is an absolute value. Of course, a zero tolerance is not possible to maintain and may cause manufacturing to question the sanity of the designer that specifies zero tolerances.

Another precaution is to not specify small size tolerances when using the zero position tolerance at MMC. A zero tolerance at MMC is intended



$\varnothing .250 \begin{smallmatrix} +.038 \\ -.000 \end{smallmatrix}$
 $\boxed{\varnothing .000 \text{ (M) A B C}}$

	Produced hole diameter	Allowable position tolerance
MMC	.250	.000
	.251	.001
	.252	.002
	.281	.031
	.282	.032
	.283	.033
LMC	.287	.037
	.288	.038

Figure 12-21. Properly applying zero position tolerances at MMC can increase manufacturing freedom and permit acceptance of all functionally acceptable holes.

to increase manufacturing freedom by placing the position tolerance and size tolerance into one value. This lets those in manufacturing determine how to distribute the tolerance within their manufacturing methods.

Position Tolerance at LMC

Distance between the edge of a part and a hole is sometimes a major consideration in the allowable location of a hole. Position tolerances may be specified with the least material condition (LMC) modifier when the remaining material outside the hole is a primary concern. See **Figure 12-22**.

The LMC modifier indicates that the specified tolerance value applies when the controlled feature is at its least material condition. In the case of a hole, the least material condition is when the

hole is at its maximum diameter. A position tolerance specified to apply at LMC will increase as the controlled feature departs from its LMC size.

The given figure shows a hole located by a basic dimension of .500" from the edge of the part. When the hole has a least material condition of .510" diameter, the allowable position tolerance is .040" diameter. A hole produced at a diameter of .510" and at the maximum displacement allowed by the .040" diameter position tolerance has an edge distance of .225".

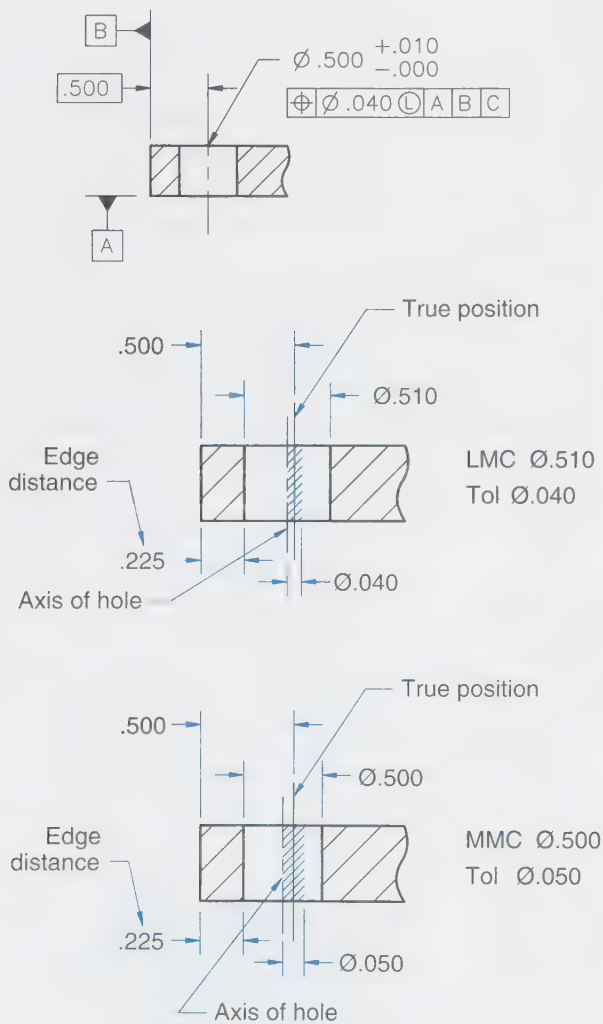
A hole produced at the minimum allowable diameter of .500" is permitted a position tolerance of .050" because of the LMC modifier. A hole produced at the .500" diameter and at the maximum displacement allowed by the .050" diameter position tolerance has an edge distance of .225".

The minimum edge distance calculated for the least material condition is maintained when using the LMC modifier. The two limits of size for the specified hole in the figure show that the edge distance is maintained at both limits when the maximum allowable position tolerance is used.

Allowable position tolerance at the MMC size must be considered when sizing holes for installation of fasteners. If the LMC modifier is applied to a tolerance specification, the allowed tolerance at MMC must also be calculated. The allowable tolerance at MMC must be a value that will work with the size hole and fastener being used. The hole size and position tolerance value must be adjusted to result in the desired minimum edge distance and also permit fastener installation.

Paper Gaging Techniques

Paper gaging is a process of recording inspection data on paper to determine if the measured part meets drawing specifications. It is useful for demonstrating how the position tolerances may be verified, and the procedure is sometimes used when coordinate measurements for hole positions are obtained during inspection. It is also acceptable to use functional gages to verify that one or more features are in compliance with tolerance requirements, but functional gages can be expensive to design and fabricate. Inspection is commonly done using computer-driven measurement machines or measurement devices that are used on the fabrication machines. Computer programs used with those machines assess the measurement data to determine compliance with tolerance requirements, and in effect complete something similar to the paper gaging process, except it is done mathematically.



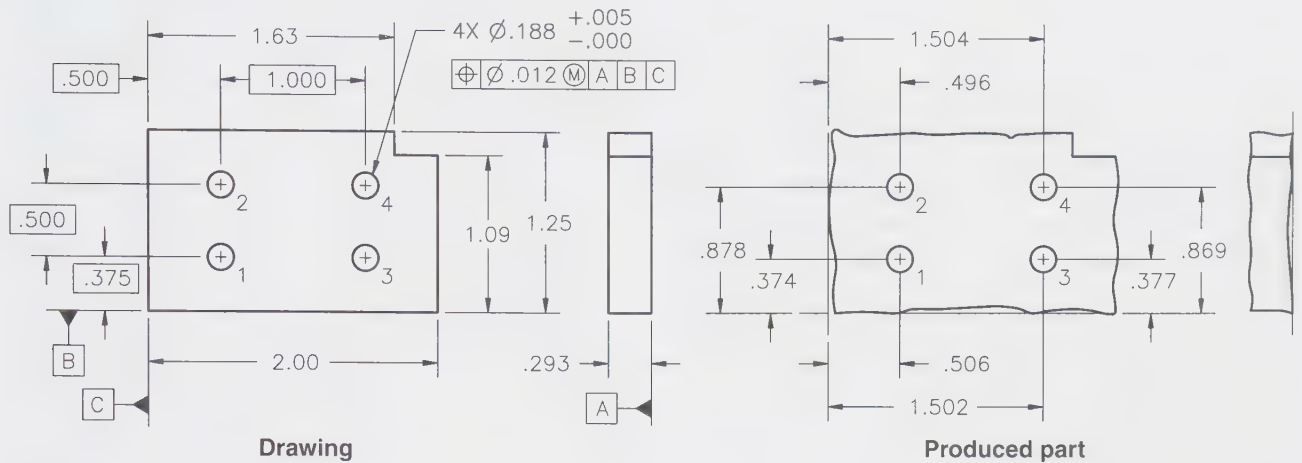
Goodheart-Willcox Publisher

Figure 12-22. Edge distance can be controlled to one minimum value by using the LMC modifier on hole position tolerances.

Verifications of hole size and location are both required to determine if drawing requirements are met, and the following examples show how paper gaging would be used. Understanding the paper gaging process also helps in understanding the

position tolerance requirements. See [Figure 12-23](#). To perform the paper gaging process, some measurement data must be obtained.

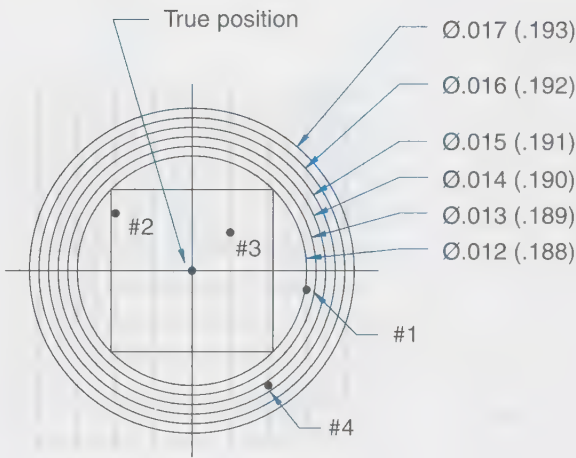
Hole size can initially be checked for compliance with size limits by using go/no-go gages.



Hole #	1		2	
Diameter	.191		.192	
	X	Y	X	Y
Measured location	.506	.374	.496	.878
Drawing dimension	.500	.375	.500	.875
Variation	.006	-.001	-.004	.003

Hole #	3		4	
Diameter	.191		.191	
	X	Y	X	Y
Measured location	1.502	.377	1.504	.869
Drawing dimension	1.500	.375	1.500	.875
Variation	.002	.002	.004	-.006

Measured hole data



Plotted coordinate variations and position tolerance zones

Mathematical solution

Hole #	Coordinate variation		Position variation	Bonus Size-MMC	Allowable position	Accept Reject
1	.006	-.001	.0122	.003	.015	Accept
2	-.004	.003	.0100	.004	.016	Accept
3	.002	.002	.0056	.003	.015	Accept
4	.004	-.006	.0144	.003	.015	Accept

Goodheart-Willcox Publisher

Figure 12-23. Paper gaging of position tolerances is one method of verifying whether or not measured location variations are acceptable.

These gages determine if a hole is within the allowable size range, but they do not determine the actual hole size. The actual hole size may need to be known if it becomes necessary to take advantage of the additional position tolerance that can be allowed based on departure from MMC.

Hole locations relative to the datum reference frame are required. Failure to measure hole locations relative to the datum reference frame is likely to result in inaccurate paper gaging conclusions. To assist with the paper gaging method, location measurements for all holes are recorded in a table. See [Figure 12-23](#). From these measured values, the coordinate variations are calculated. Care must be taken to accurately determine whether the variation values are positive or negative.

Coordinate variation values may be used to calculate the diameter position variations, or they may be plotted on a grid to determine the diameter position variations. The given figure shows both methods.

Coordinate variations are plotted on a grid at a greatly enlarged scale. A minimum scale of 100:1 is recommended. A scale of 100:1 allows using a .100" grid to represent .001" of variation.

Variations calculated in the shown table are plotted on the grid. A point is located for each hole and the associated hole identification number is placed by the point. After all points are plotted, a set of concentric circles is centered on the origin for the grid. A scale equal to that of the grid must be used to draw the circles. The smallest circle is drawn to represent the smallest permitted position tolerance. Larger circles are drawn at an increment to represent .001" diameter.

Each of the concentric circles represents a tolerance zone diameter, and each tolerance zone diameter is associated with a hole size. Assuming the position tolerance is specified with an MMC modifier, the smallest diameter circle (representing .012" diameter) is associated with the MMC size (.188" diameter) of the hole. The next larger tolerance diameter circle is associated with a feature that has departed from the MMC value by .001". The second circle represents a .013" diameter tolerance zone. It is associated with a hole size of .189" diameter.

Any hole produced at a .188" diameter must have a position variation within a .012" diameter. Any hole produced at a .189" diameter must have a position variation within a .013" diameter.

The given figure shows four points plotted on the grid in positions representing hole locations. Concentric circles located on the grid origin show the position variation of each hole. Holes #2 and

#3 are both within the smallest allowable position tolerance; therefore, their actual hole size is unimportant, provided the hole diameters are somewhere within the allowable size tolerance.

Hole #1 is located outside the .012" diameter circle, but inside the .013" diameter circle. This means that this hole must have a diameter of at least .189". The hole size must be checked if it is not already recorded in the inspection data table. In the figure, hole #1 is shown to be .191" diameter. It is therefore acceptable in the shown location. In a similar way, hole #4 is determined to be acceptable.

An alternate to paper gaging is to mathematically determine if the hole positions are acceptable. See the table at the bottom of [Figure 12-23](#). The coordinate variations are used to determine the diameter of positional variation either by calculation or using the table in Appendix A of this textbook. Bonus tolerance is determined by finding the difference between the produced hole diameter and the specified MMC. Allowable position is equal to the specified tolerance plus the bonus tolerance. The hole position is acceptable when the position variation is less than the allowable position.

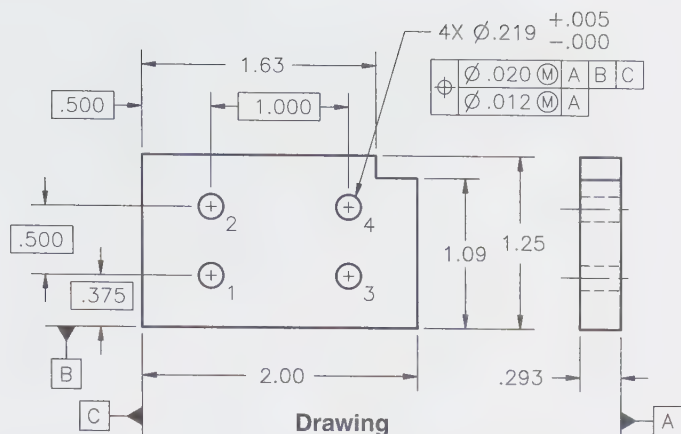
Paper gaging techniques may also be applied to composite position tolerances. The pattern-locating tolerances are verified in the same manner as described for the single-segment position tolerance feature control frame. The feature-relating tolerance can be verified in a slightly different manner.

[Figure 12-24](#) shows a paper gaging technique for composite position tolerances. The pattern-locating tolerance for the given tolerance specification is verified in the manner described for the previous figure.

The feature-relating tolerance shown in the second segment of the composite tolerance specification is verified by measuring the relative locations of the holes. Measurements are made in a plane parallel to primary datum A, because datum A is referenced in the second line of the tolerance specification. Measurements between the holes are typically made with one of the holes acting as an origin. In the given figure, hole #1 is selected to serve as the origin.

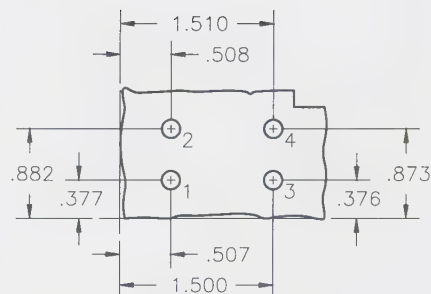
Direction of measurements is established by selecting a second hole. Hole #3 in the given figure is selected for this purpose. After two holes are selected to locate the origin and orient the coordinate system axes, all hole locations are measured from the coordinate system that is located by the two selected holes.

Measurements for the relative locations of the holes are recorded in a table and coordinate



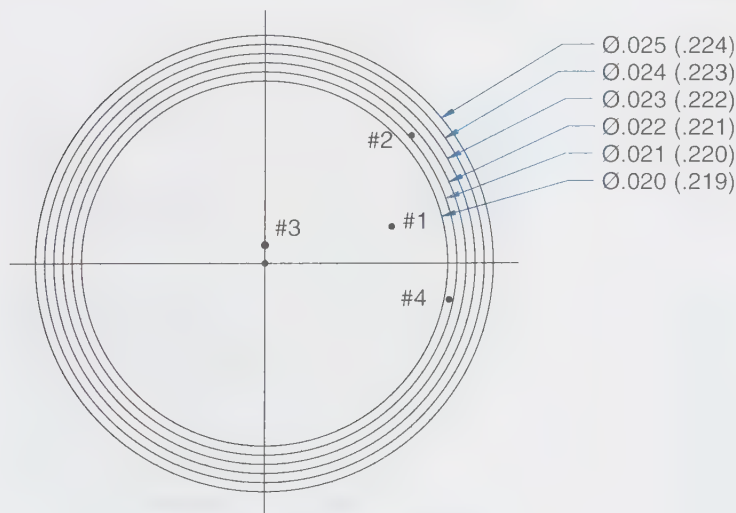
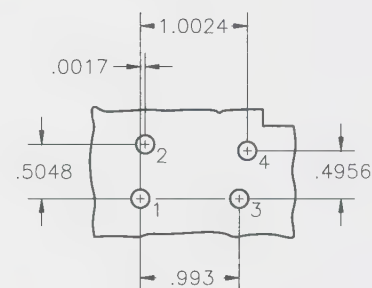
Hole location coordinate variations from datums

Hole #	1		2		3		4	
Diameter	.222		.223		.221		.223	
	X	Y	X	Y	X	Y	X	Y
Measured location	.507	.377	.508	.882	1.500	.376	1.510	.873
Drawing dimension	.500	.375	.500	.875	1.500	.375	1.500	.875
Variation	.007	.002	.008	.007	.000	.001	.010	-.002

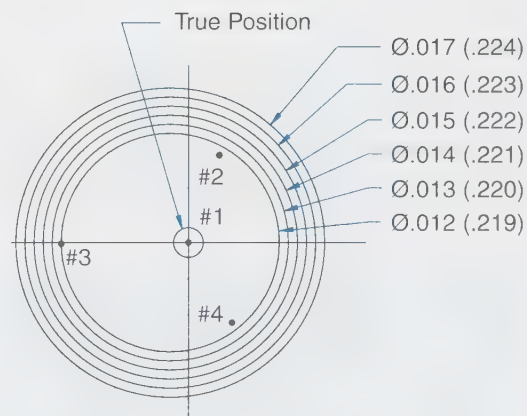


Hole to hole location variations

Hole #	1		2		3		4	
Diameter	.222		.223		.221		.223	
	X	Y	X	Y	X	Y	X	Y
Measured location	0	0	.0017	.5048	.9930	0	1.0024	.4956
Drawing dimension	0	0	0	.500	1.000	0	1.000	.500
Variation	0	0	.0017	.0048	-.0070	0	.0024	-.0044



Position relative to datums



**Hole to hole relative positions
(feature to feature)**

Goodheart-Willcox Publisher

Figure 12-24. Composite position tolerances may be paper gaged by separately plotting hole locations as measured relative to the true positions created by the pattern-locating and the feature-relating tolerances.

variations determined. Whether the coordinate variations are positive or negative is important and must be accurately recorded.

Coordinate variations are plotted on a grid. Care must be taken to plot the points in the proper locations. Positive and negative values must be properly plotted to obtain accurate results.

After all points are plotted, a set of concentric circles representing the allowable feature-relating tolerances and the associated feature sizes is placed on the grid. The concentric circles may be placed in any position that encloses the plotted points. There is no requirement to center the circles on the origin of the grid. Forcing the circles to be centered on the grid would be incorrect because this would be the same as constraining translation (fixing location) of the tolerance zones relative to the datum reference frame. There is no requirement for the feature-relating tolerance zones to be constrained in translation relative to the datum reference frame.

Should the holes be slightly outside the allowed feature-relating tolerance, the results may improve if different holes are used to establish the origin and stop rotation of the coordinate system. However, the results will not improve enough to matter if the feature-to-feature variation exceeds the specified tolerance.

Feature-Relating Tolerance Quick Check (For Acceptance Only)

It is possible to make a quick acceptance check of the feature-relating tolerance before measuring the feature-to-feature positions. This procedure may only be used to accept parts and must not be used to reject parts that fail this quick check. Many good parts will pass this test, but some good parts will not pass this test. If a part passes this check method, it is not necessary to measure feature-to-feature positions.

Pro Tip

This quick check method should not be used to reject parts, because some good parts will not pass this test. If the cost associated with rejecting some good parts is acceptable, then this check method may be used for acceptance and rejection.

One set of hole location measurements is taken relative to the referenced datums for the pattern-locating tolerance, and the hole locations

are plotted on a grid as illustrated in [Figure 12-24](#). Two sets of concentric circles are then used on this one set of plotted points. The first set of circles is sized to the pattern-locating tolerances. These circles must be centered on the origin of the grid. The second set of circles is sized to the feature-relating tolerances. These circles may be located in any position encompassing all the plotted points. The feature-relating tolerance circles will only encompass all the points if the entire hole pattern has shifted (translated) in one direction but not rotated relative to the datums.

If both sets of circles enclose the points, then the hole locations are acceptable. In this case, the quick check has saved the time that would have otherwise been used to measure hole-to-hole locations.

If the pattern-locating tolerance circles fail to enclose all the points, the part is not acceptable. Depending on the variations, it may be possible to rework the part.

If the feature-relating tolerance circles fail to encircle the points, it does not mean that the part is bad. It only means that the feature-to-feature positions must be measured and plotted as illustrated in [Figure 12-24](#).

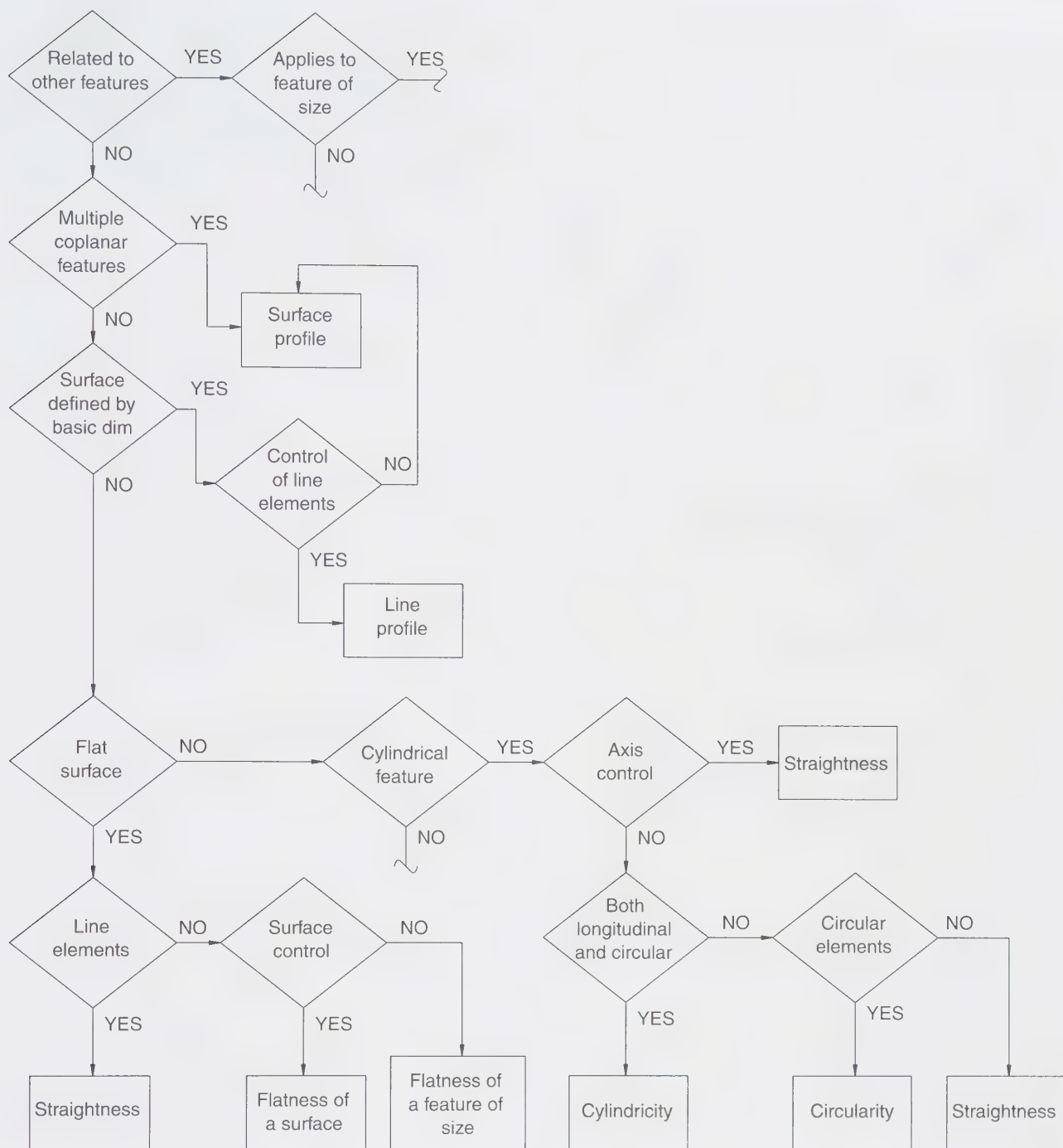
Selection of the Applicable Tolerance and Modifiers

Complex designs are usually a compilation of many relatively simple features. Application of tolerances can be made easier if the design is mentally broken down into the simple features and controls placed on them in the simplest possible means.

Once a feature is selected for application of a tolerance, it is necessary to determine which type or types of tolerances to apply. See [Figure 12-25](#). Asking questions about the required level of control for a feature is a good method to use for determining what tolerances to apply. The figure shows a flow diagram with example questions that must be asked to determine what tolerance types to apply.

Typically, the first thing that must be determined is whether or not the feature needs to be controlled relative to other features. If no relationship to other features is required, then the tolerance type is going to be some type of form or profile tolerance. Additional questions will determine exactly what tolerance type is needed.

The given figure shows a flow diagram that permits selection of a tolerance control for many applications that require controls not referenced to datums. It shows the type of process that should be performed to select the proper tolerance type. The logical process shown can be expanded to include



Goodheart-Willcox Publisher

Figure 12-25. A complete flow diagram for determining tolerance application may be created using the format shown in this figure.

the selection of tolerances for related features, the selection and referencing of datum features, and also the application of material condition modifiers.

The logic used to develop the shown flow diagram is explained in the following paragraphs. This explanation excludes groups of features to which position tolerances may be applied without a datum feature reference.

Does the feature need to be controlled relative to any other feature? If the answer is no, then the tolerance must either be a form tolerance or a profile tolerance. Form tolerances do not reference other features (datums) and profile tolerances may be used without datum feature references to control the form of a feature. The NO path from this question leads to form and profile tolerances.

Is the required control applied to multiple coplanar features? If the answer is YES, all form tolerances are eliminated because form tolerances are applied to individual features. A NO answer leads to possible form or profile tolerances. Profile tolerances may be used on individual features or multiple coplanar features.

If the control is applied to an individual feature, a question is asked to determine if the feature profile is defined by basic dimensions. If the answer is YES, the required control is profile. If NO, the control is most likely a form tolerance. Additional questions are asked to determine the type of form tolerance to apply to the feature.

Multiple-Segment Tolerance Specifications

Many tolerance types automatically include controls obtained by other tolerance types. As an example, perpendicularity applied to a flat surface also controls the flatness of the surface. If a perpendicularity tolerance of .015" is applied to a surface, that surface must also be flat within .015". It could not be perpendicular within .015" if it had a flatness variation greater than that value.

When applying a tolerance, all controls imposed by the specification should be considered. Sometimes, one specification can achieve all the needed control. Placement of unnecessary specifications on a drawing should be avoided. There is no need to apply a perpendicularity tolerance of .015" on a surface and also apply a flatness tolerance specification of .015". The flatness requirement is already included in the perpendicularity tolerance.

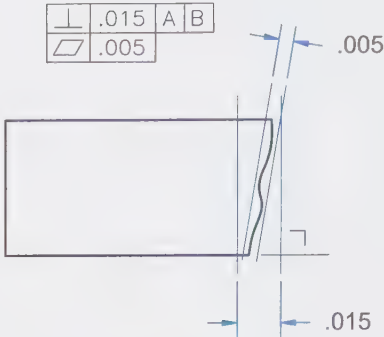
It is also important not to specify an overly restrictive, high-level control to avoid the specification of another tolerance. If a surface needs a flatness tolerance of .005", a perpendicularity tolerance of .005" should not be applied to obtain the needed flatness if a perpendicularity tolerance of .015" is good enough. It is better to specify only the needed level of perpendicularity control and specify a separate flatness requirement.

Orientation and Form on Surfaces

Multiple tolerance specifications may be applied to one feature. See Figure 12-26. A perpendicularity tolerance of .015" and a flatness tolerance of .005" are shown in the given figure. The two tolerances applied to one surface control orientation to the .015" value, and form within the .005" value. The surface to which these tolerances are applied must meet both requirements.

Orientation and Form on Features of Size

A similar specification may be applied to a hole. See Figure 12-27. A perpendicularity tolerance of .017" RFS is applied to a hole. Perpendicularity applied RFS controls the axis of the hole. The

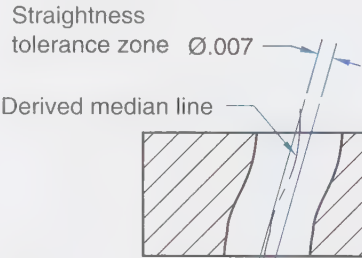


Goodheart-Willcox Publisher

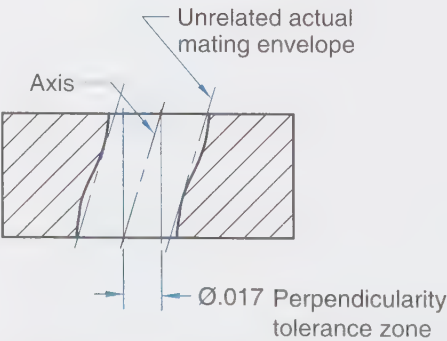
Figure 12-26. One tolerance specification may be used to refine the requirements of a higher level tolerance specification.



Tolerance Specification



Straightness Requirement



Perpendicularity Requirement

Goodheart-Willcox Publisher

Figure 12-27. Tolerances applied to features of size may also include multiple levels of control.

hole also has a derived median line straightness tolerance of .007" RFS. The straightness tolerance applied RFS controls the derived median line. These are two different derived features. The axis is a straight line. The derived median line follows the shape of the hole and may not be straight.

The axis (of the unrelated actual mating envelope) must be within a perpendicularity tolerance zone that is .017" diameter, and the derived median line must be within a straightness tolerance zone that is .007" diameter.

Because the perpendicularity tolerance applied RFS controls the axis of the unrelated actual mating envelope and the straightness tolerance applied RFS controls the derived median line, there are two different derived features being controlled. It is difficult to say that one controls the other, but there are differences of opinion in industry regarding whether the straightness tolerance must be contained within the orientation tolerance. This question only arises when the tolerances are applied to features of size. If the two tolerances are evaluated separately, whether one is contained by the other is not important.

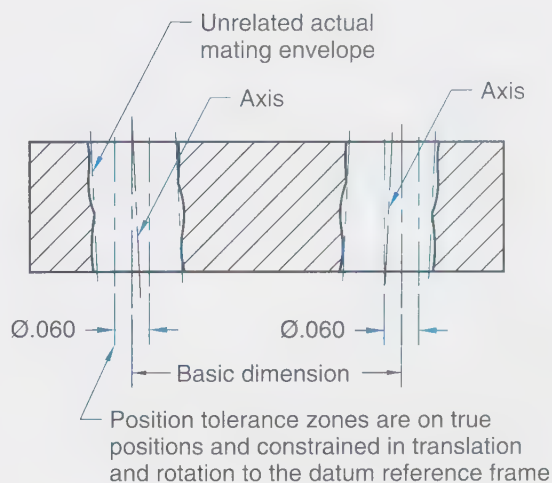
Position, Orientation, and Form

Application of a position tolerance should only require the level of control needed for position. If further refinement of feature orientation or form is needed, then that control should be achieved with the correct tolerance specification, not with overly restrictive position tolerances. See [Figure 12-28](#). The given figure shows an allowable position tolerance of .060" diameter applied RFS. An orientation tolerance of .012" diameter is applied RFS to the same feature to prevent the orientation variation from being the full .060" permitted by the position tolerance. A straightness tolerance of .006" diameter is also applied RFS to the feature to further control the form of the feature.

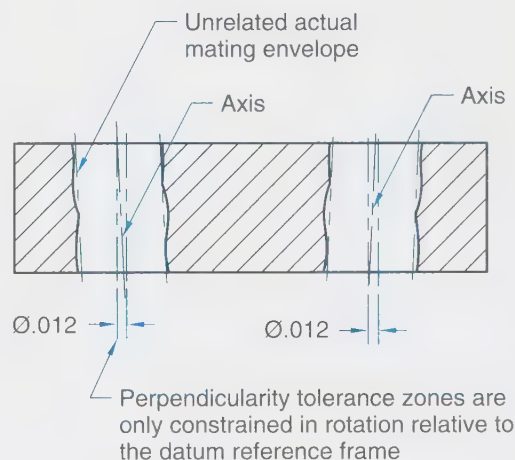
The position tolerance applied RFS controls the axis of the unrelated actual mating envelope, as does the perpendicularity tolerance. Because they both control the same derived feature, it is known that the perpendicularity is also controlled by the position tolerance. Again, the straightness tolerance applied RFS controls the derived median line. As explained earlier, the straightness tolerance is controlling a different derived feature. Therefore, it is not logical to assume that the straightness is also controlled by the other tolerances. Once again, verifying the tolerances separately confirms whether each tolerance is met, and there is no need to debate whether one tolerance is containing the other.

\oplus	$\varnothing .060$	A	B	C
\perp	$\varnothing .012$	A		
$-$	$\varnothing .006$			

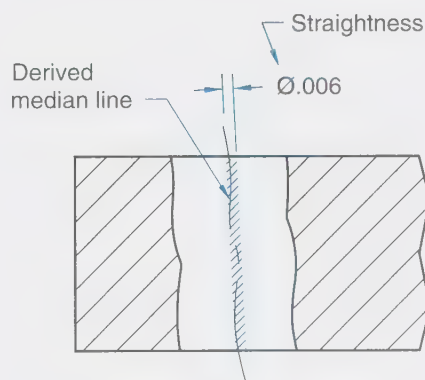
Tolerance Specification



Position Requirement



Perpendicularity Requirement



Straightness Requirement

Goodheart-Willcox Publisher

Figure 12-28. Only the level of control required to make the part functionally acceptable should be applied to any feature.

Multiple levels of control should be used when they increase producibility of the parts. However, multiple levels of control should be applied with caution. The amount of effort necessary to verify two or three tolerance requirements must be balanced against the effort required to meet one tolerance specification that achieves all the needed controls within one specified tolerance value.

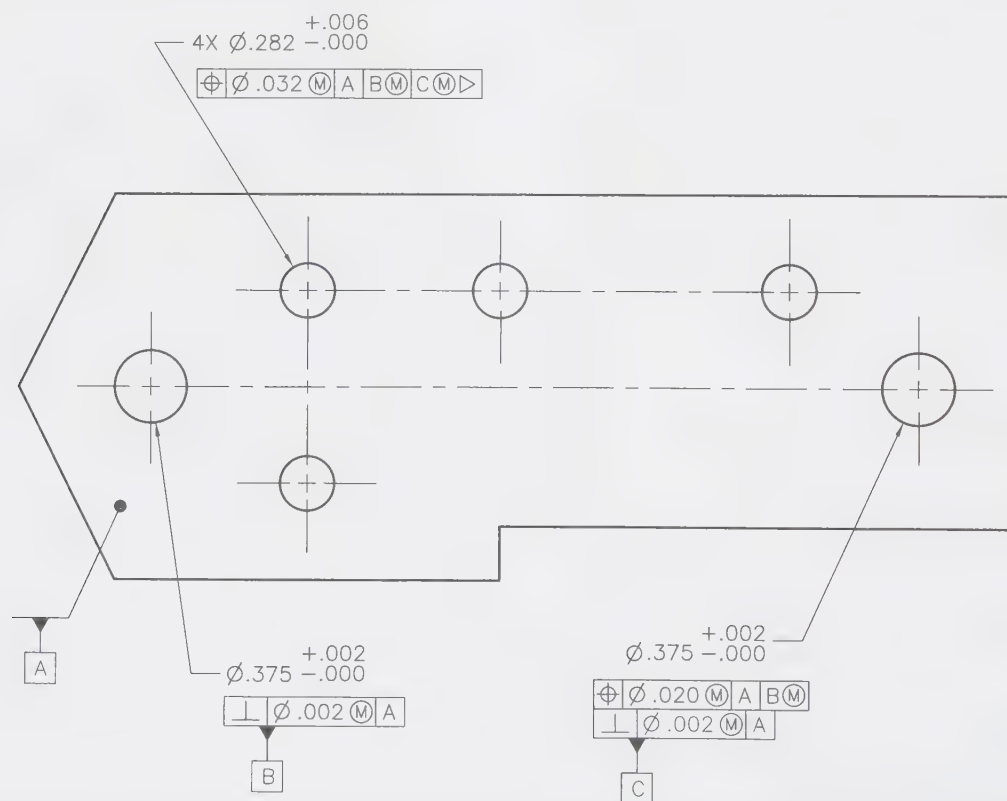
Effects of Tolerances Applied at MMC in Combination with Datum References at MMB

A question often arises regarding how additional tolerance is gained when a datum feature of size departs from its MMB. Additional tolerances resulting from the MMC modifier were explained for each of the tolerances where the modifier may be used. The potential for additional tolerance (that is not directly added) resulting from the MMB modifier on datum feature references was explained in the chapter on datums. The following explanation shows the possible combined effects of a tolerance with the MMC modifier and one or more datum references with the MMB modifier. These modifiers affect the allowable variation on the detail part, and that variation has the potential to affect an assembly that uses the part.

A position tolerance may be applied with the MMC modifier on the tolerance value and the MMB modifier on references to datum features of size. See [Figure 12-29](#). There are four .282" diameter holes that have a position tolerance of .032" applied MMC, with references to secondary datum feature B at MMB and tertiary datum feature C at MMB with the translation modifier. The material condition modifiers and the material boundary modifiers have the potential to result in allowable variation on the hole positions that is in addition to the shown tolerance value. Not all the allowable increase is through simple addition. There are geometry effects that must be considered.

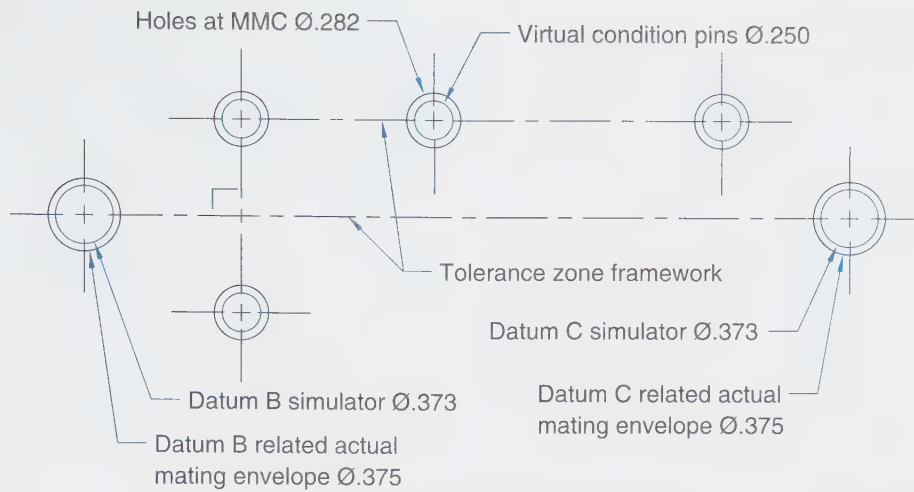
The departure from MMC of the .282" diameter holes does add directly to the specified position tolerance on those holes as has been previously explained. For every .001" increase in the size of a hole, that hole gets additional tolerance of .001" diameter.

The effects of datum feature size can be seen through observing the possible movement of a datum simulator within the datum feature. See [Figure 12-30](#). The first segment of the figure shows a gage with datum simulators and virtual condition pins on the true positions within the tolerance zone framework for the given part in

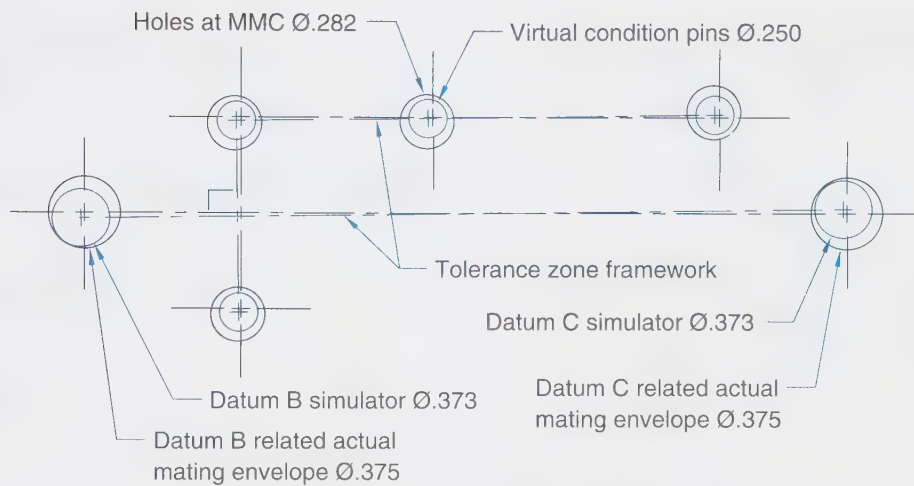


Goodheart-Willcox Publisher

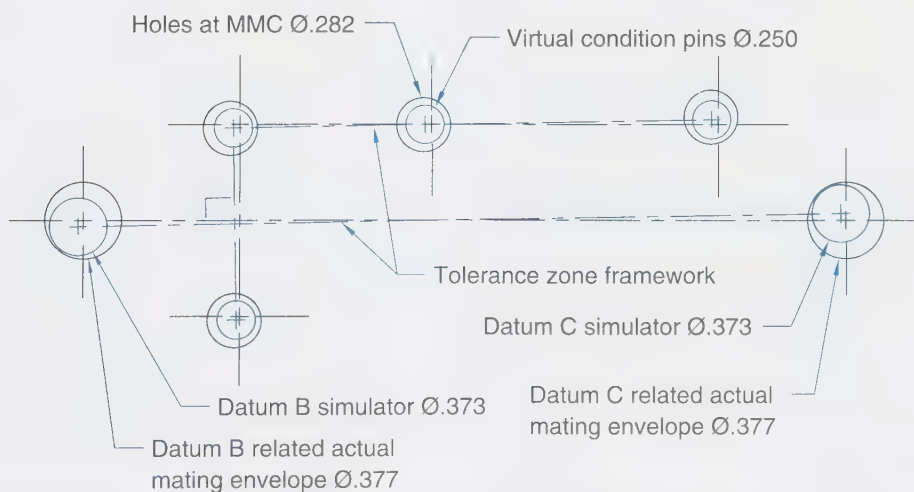
Figure 12-29. Additional tolerance can be available when datum features are referenced MMB.



Gage Pins and Holes at True Position



Allowable Gage Movement with Datum Feature Holes at MMC



Allowable Gage Movement with Datum Feature Holes at LMC

Goodheart-Willcox Publisher

Figure 12-30. The combined effects of position tolerances and datum feature size on the hole locations established for the part in Figure 12-29.

Figure 12-29. All produced holes are shown in true positions at their MMC sizes. The datum feature holes are shown with no variation in them, so there is clearance between the datum features and the simulator pins. There is also clearance between the virtual condition pins and the .282" diameter holes. It can be seen that the clearance between the datum features and the datum simulators of the gage will accommodate some movement of the pattern of holes. In effect, the clearance is allowing the tolerance zone framework to move. In the given example, there is only .002" clearance between the datum simulator and the datum feature when the datum feature is at MMB.

The second segment of the given figure shows variation in the hole locations. All holes are at the MMC size, including the datum feature holes, which are perpendicular to Datum A. The gage is shown moved into a position that allows all virtual condition pins to fit into the holes. The amount of movement of the gage is limited by the clearance between the datum simulator pins and the datum feature holes. In this case, the gage has translated (shifted) and slightly rotated.

The last segment of the figure shows the datum feature holes at their largest permitted size (LMC). The gage is permitted to translate and rotate using the increased clearance. In this case, the gage is shown moved in a direction that accepts all the holes that have a slightly increased amount of location variation.

The departure of the datum feature holes from their MMB does not result in a direct increase in the position tolerances on the .282" diameter holes. If datum feature B departs from its MMB, the tolerance zone framework for all the holes can move, but the tolerance on individual holes is unchanged. In effect, the origin for the holes is allowed to move by an amount equal to the additional radial clearance in the datum B hole. Similarly, if the datum feature C hole departs from its MMB, the tolerance zone framework is permitted an increased amount of rotation around the origin at datum B.

However, the tolerance zone framework must be in one location and orientation for checking all the features that reference the datums. The framework is not permitted to be in one location for one of the tolerated holes, and move to another location for other holes.

The purpose of the specified tolerance is to ensure the hole pattern will be accurate within the tolerance value. The datum references constrain the location of the hole pattern, and using

the MMB modifier allows the location of the entire pattern some movement relative to the referenced datums.

The assembly effects of the above described variation should be evaluated. Both the increased diameters of the holes (that allowed the additional tolerance), and the additional location variation that is allowed by the departure from MMC and MMB, will combine to permit location variation of parts in an assembly. As was shown in the above explanation, the true position for the pattern of holes was permitted to move relative to the referenced datums, and that movement will result in a mating part that is moved from the basic location. Also, the hole diameters have departed from MMC, so there is more clearance for the mating fasteners to move.

It is sometimes necessary to specify datum references at RMB to prevent the assembly level variation that can result from datum feature references made at MMB.

Chapter Summary

- ✓ The total allowable tolerance for both the floating fastener and fixed fastener conditions may be distributed for holes of one or more sizes.
- ✓ Tolerances for a floating fastener condition with three parts containing equal-size holes stacked together may be calculated in the same manner as when two parts are stacked together.
- ✓ Tolerances for a fixed fastener condition with three stacked parts may be calculated considering the parts two at a time, provided the fixed fastener feature is included in the calculations for each pair of parts.
- ✓ When tolerances are distributed unevenly or fixed fastener conditions exist, projected tolerance zones or orientation tolerances are needed to prevent interference conditions.
- ✓ Tolerance accumulation may be minimized through proper selection of datum features.
- ✓ Zero position tolerances applied at MMC can increase manufacturing freedom and reduce rejection of functionally good parts.
- ✓ Position tolerances including the LMC modifier are sometimes used to control edge distances.
- ✓ Paper gaging techniques permit manually obtained inspection data to be evaluated for compliance with position tolerances.

Review Questions

Answer the following questions using a separate sheet of paper for your answers. Additional review questions and problems are available with ample workspace in the Study Guide.

Multiple Choice

- When applying a position tolerance to a threaded hole, a ____ tolerance zone is typically used to prevent interference in the mating clearance hole.
 - small
 - large
 - profile
 - projected
- ____ datum features are identified on both parts and referenced in position tolerances when alignment of edges in an assembly is required.
 - Random
 - Similar
 - Unrelated
 - No
- Selection of the proper ____ can reduce tolerance accumulation.
 - datum features
 - hole sizes
 - fastener sizes
 - None of the above.
- The position tolerances applied to the holes in a floating fastener application ____ applied to each of the parts.
 - may be one value
 - may be two individual tolerances
 - must be one value
 - Both A and B.
- Stacking three parts that all contain clearance holes of one size requires completion of calculations ____.
 - using the same formula as is used for two parts
 - using the same formula as for fixed fastener applications
 - considering only two parts at a time
 - None of the above.
- The ____ datum feature reference in a position tolerance specification on holes is typically the datum feature that makes contact with the attached part.
 - primary
 - secondary
 - tertiary
 - Either A or C.
- Datum features of parts in an assembly ____ if a single-segment position tolerance is specified and the tolerances are calculated as explained in this chapter.
 - will not align
 - will align
 - will offset by the value of the tolerance
 - None of the above.
- The ____ tolerance in a composite position tolerance controls relative location of holes in a group and ensures the fasteners can pass through the holes.
 - pattern-locating
 - feature-relating
 - group-holding
 - MMC-specified
- The ____ tolerance in a composite tolerance most directly affects the amount of misalignment between features in an assembly when the parts are bolted together.
 - pattern-locating
 - feature-relating
 - group-holding
 - MMC-specified
- Control of ____ sometimes warrants the use of the LMC modifier on position tolerances.
 - assembly misalignment
 - floating fastener conditions
 - fixed fastener conditions
 - edge distance

True/False

- True or False?* The formula $T = H - F$ may only be used to complete calculations when a maximum of two parts is stacked together.

12. *True or False?* If a .010" diameter position tolerance is used on two stacked parts, the clearance hole diameter for a fixed fastener condition must be larger than the clearance hole diameter for a floating fastener condition.
13. *True or False?* It would be extremely restrictive, if not impossible to meet, a tolerance of .000" RFS.

Fill in the Blank

14. A(n) _____ position tolerance at MMC in combination with a large size tolerance can increase manufacturing freedom when properly utilized.
15. A projected tolerance zone is normally specified to prevent _____ between the fastener and the clearance hole in a fixed fastener application.
16. _____ are used to represent position tolerance zones when paper gaging.
17. Paper gaging should be completed at a recommended scale of at least _____:1.

Short Answer

18. List the two general fastener conditions for which position tolerances must be calculated.
19. Describe a floating fastener condition.
20. Show the formula for a fixed fastener condition in which two parts are stacked together and a projected tolerance zone is to be used on the fixed feature.
21. What is the formula for calculating position tolerances for a floating fastener condition in which there are two hole sizes and the tolerances are to be distributed unevenly?

Application Problems

Each of the following problems requires that calculations be completed. Show your calculations.

- 22–25. Use the data from the given table to calculate a position tolerance for each problem. Assume that two parts are stacked together and that the tolerance is for a floating fastener condition.

Problem number	Fastener MMC	Hole diameter at MMC	
		Part #1	Part #2
22	Ø.164	Ø.193	Ø.193
23	Ø.190	Ø.218	Ø.218
24	Ø.190	Ø.226	Ø.226
25	Ø.250	Ø.282	Ø.282

- 26–29. Use the data from the given table to calculate a position tolerance for each problem. Assume that two parts are stacked together and that the tolerance is for a fixed fastener condition. Assume that a projected tolerance zone is used.

Problem number	Fastener MMC	Hole MMC
26	Ø.164	Ø.188
27	Ø.190	Ø.212
28	Ø.250	Ø.282
29	Ø.250	Ø.312

- 30–32. Use the data from the given table to calculate a clearance hole diameter for each problem. Assume that two parts are stacked together and that a fixed fastener condition exists. Assume that a projected tolerance zone is used.

Problem number	Fastener MMC	Position tolerance
30	Ø.138	Ø.018
31	Ø.190	Ø.024
32	Ø.250	Ø.029

Coordinate-to-Diameter Conversion Table

Example:

$$X = .005''$$
$$Y = 007''$$

The required diameter position tolerance zone to permit this measured location is .01720".

Conversion Table

		X Coordinate																				
		.000	.001	.002	.003	.004	.005	.006	.007	.008	.009	.010	.011	.012	.013	.014	.015	.016	.017	.018	.019	.020
.000	Y Coordinate	.00000	.00200	.00400	.00600	.00800	.01000	.01200	.01400	.01600	.01800	.02000	.02200	.02400	.02600	.02800	.03000	.03200	.03400	.03600	.03800	.04000
.001		.00200	.00283	.00447	.00632	.00825	.01020	.01217	.01414	.01612	.01811	.02010	.02209	.02408	.02608	.02807	.03007	.03206	.03406	.03606	.03805	.04005
.002		.00400	.00447	.00566	.00721	.00894	.01077	.01265	.01456	.01649	.01844	.02040	.02236	.02433	.02631	.02828	.03027	.03225	.03423	.03622	.03821	.04020
.003		.00600	.00632	.00721	.00849	.01000	.01166	.01342	.01523	.01709	.01897	.02088	.02280	.02474	.02668	.02864	.03059	.03256	.03453	.03650	.03847	.04045
.004		.00800	.00825	.00894	.01000	.01131	.01281	.01442	.01612	.01789	.01970	.02154	.02341	.02530	.02720	.02912	.03105	.03298	.03493	.03688	.03883	.04079
.005		.01000	.01020	.01077	.01166	.01281	.01414	.01562	.01720	.01887	.02059	.02236	.02417	.02600	.02786	.02973	.03162	.03353	.03544	.03736	.03929	.04123
.006		.01200	.01217	.01265	.01342	.01442	.01562	.01697	.01844	.02000	.02163	.02332	.02506	.02683	.02864	.03046	.03231	.03418	.03606	.03795	.03985	.04176
.007		.01400	.01414	.01455	.01523	.01612	.01720	.01844	.01980	.02126	.02280	.02441	.02608	.02778	.02953	.03130	.03311	.03493	.03677	.03863	.04050	.04238
.008		.01600	.01612	.01649	.01709	.01789	.01887	.02000	.02126	.02263	.02408	.02561	.02720	.02884	.03053	.03225	.03400	.03578	.03758	.03940	.04123	.04308
.009		.01800	.01811	.01844	.01897	.01970	.02059	.02163	.02280	.02408	.02546	.02691	.02843	.03000	.03162	.03329	.03499	.03672	.03847	.04025	.04205	.04386
.010		.02000	.02010	.02040	.02088	.02154	.02236	.02332	.02441	.02561	.02691	.02828	.02973	.03124	.03280	.03441	.03606	.03774	.03945	.04118	.04294	.04472
.011		.02200	.02209	.02236	.02280	.02341	.02417	.02506	.02608	.02720	.02843	.02973	.03111	.03256	.03406	.03561	.03720	.03883	.04050	.04219	.04391	.04565
.012		.02400	.02408	.02433	.02474	.02530	.02600	.02683	.02778	.02884	.03000	.03124	.03256	.03394	.03538	.03688	.03842	.04000	.04162	.04327	.04494	.04665
.013		.02600	.02608	.02631	.02668	.02720	.02786	.02864	.02953	.03053	.03162	.03280	.03406	.03538	.03677	.03821	.03970	.04123	.04280	.04441	.04604	.04771
.014		.02800	.02807	.02828	.02864	.02912	.02973	.03046	.03130	.03225	.03329	.03441	.03561	.03688	.03821	.03960	.04104	.04252	.04405	.04561	.04720	.04883
.015		.03000	.03007	.03027	.03059	.03105	.03162	.03231	.03311	.03400	.03499	.03606	.03720	.03842	.03970	.04104	.04243	.04386	.04534	.04686	.04841	.05000
.016		.03200	.03206	.03225	.03256	.03298	.03353	.03418	.03493	.03578	.03672	.03774	.03883	.04000	.04123	.04252	.04386	.04525	.04669	.04817	.04968	.05122
.017		.03400	.03406	.03423	.03453	.03493	.03544	.03606	.03677	.03758	.03847	.03945	.04050	.04162	.04280	.04405	.04534	.04669	.04808	.04952	.05099	.05250
.018		.03600	.03606	.03622	.03650	.03688	.03736	.03795	.03863	.03940	.04025	.04118	.04219	.04327	.04441	.04561	.04686	.04817	.04952	.05091	.05235	.05381
.019		.03800	.03805	.03821	.03847	.03883	.03929	.03985	.04050	.04123	.04205	.04294	.04391	.04494	.04604	.04720	.04841	.04968	.05099	.05235	.05374	.05517
.020		.04000	.04005	.04020	.04045	.04079	.04123	.04176	.04238	.04308	.04386	.04472	.04565	.04665	.04771	.04883	.05000	.05122	.05250	.05381	.05517	.05657

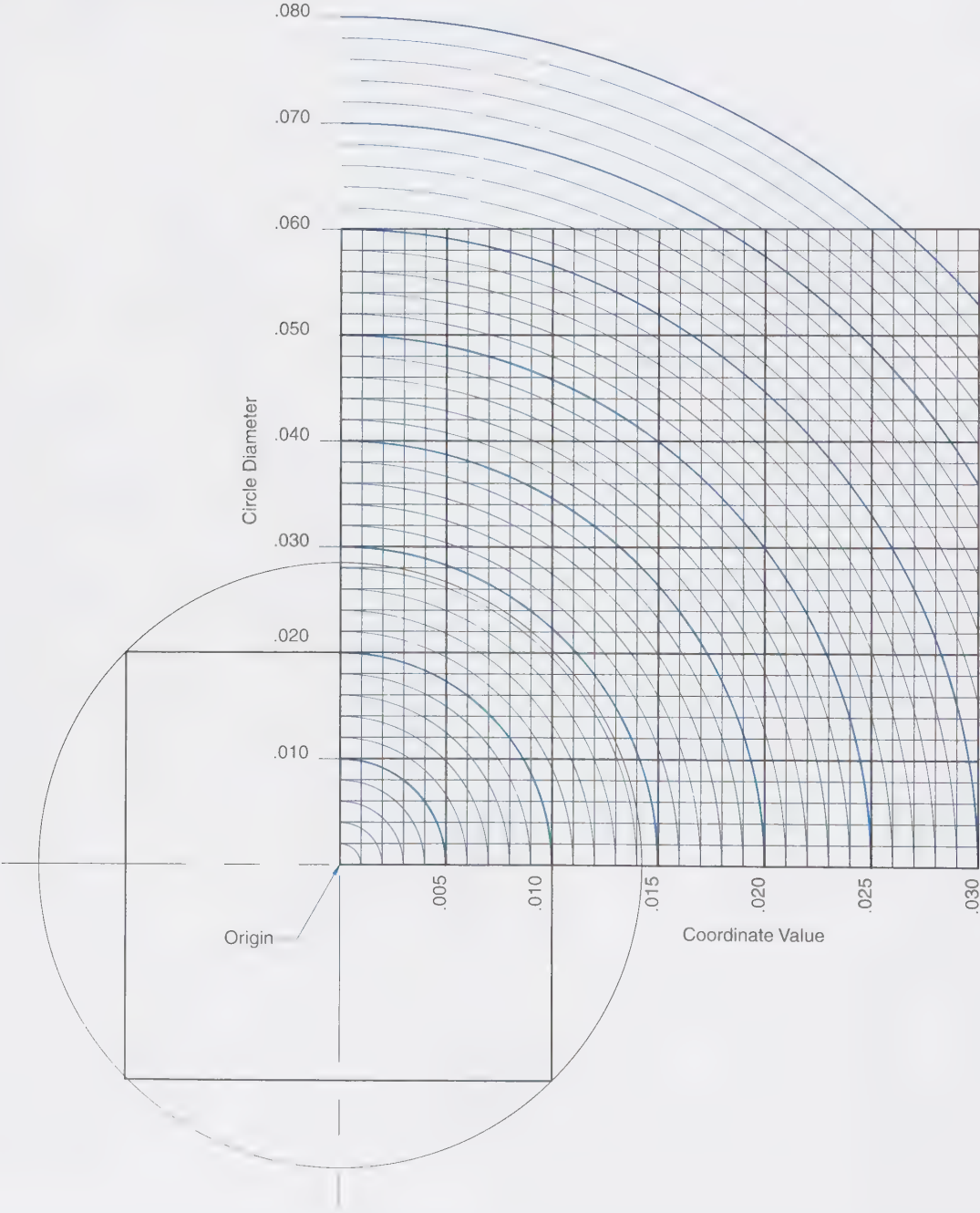
Appendix A2

Coordinate-to-Diameter Conversion Chart

This figure may be used to approximate the required diameter circle to inscribe a coordinate value. Grid lines are drawn on a .001" increment, and circles are drawn on a .002" diameter increment. Coordinate values in the X and Y axes are equal. Coordinate values are only labeled along the X axis. Diameters are labeled along the Y axis.

The chart is used by plotting a coordinate value on the grid. A circle passing through the plotted point, or the next larger circle, is traced counterclockwise to the Y axis to determine the diameter.

Any coordinate value may be plotted on this chart. If coordinate values extend beyond the chart boundaries, the scale of the chart may be increased. Simply multiply all values on the chart by the same number to increase its scale.



Appendix A3

Drill Size to Decimal Inch Diameter Table

The following table provides drill size designations and common decimal diameters.

Drill Size	Decimal Diameter
80	0.0135
79	0.0145
1/64	0.0156
78	0.0160
77	0.0180
76	0.0200
75	0.0210
74	0.0225
73	0.0240
72	0.0250
71	0.0260
70	0.0280
69	0.0292
68	0.0310
1/32	0.0313
67	0.0320
66	0.0330
65	0.0350
64	0.0360
63	0.0370
62	0.0380
61	0.0390
60	0.0400
59	0.0410
58	0.0420
57	0.0430
56	0.0465
3/64	0.0469
55	0.0520
54	0.0550
53	0.0595
1/16	0.0625
52	0.0635
51	0.0670

Drill Size	Decimal Diameter
50	0.0700
49	0.0730
48	0.0760
5/64	0.0781
47	0.0785
46	0.0810
45	0.0820
44	0.0860
43	0.0890
42	0.0935
41	0.0960
40	0.0980
39	0.0995
38	0.1015
37	0.1040
36	0.1065
7/64	0.1094
35	0.1100
34	0.1110
33	0.1130
32	0.1160
31	0.1200
1/8	0.1250
30	0.1285
29	0.1360
28	0.1405
9/64	0.1406
27	0.1440
26	0.1470
25	0.1495
24	0.1520
23	0.1540
5/32	0.1563
22	0.1570
21	0.1590

Drill Size	Decimal Diameter
20	0.1610
19	0.1660
18	0.1695
11/64	0.1719
17	0.1730
16	0.1770
15	0.1800
14	0.1820
13	0.1850
3/16	0.1875
12	0.1890
11	0.1910
10	0.1935
9	0.1960
8	0.1990
7	0.2010
13/64	0.2030
6	0.2040
5	0.2055
4	0.2090
3	0.2130
7/32	0.2188
2	0.2210
1	0.2280
A	0.2340
15/64	0.2344
B	0.2380
C	0.2420
D	0.2460
1/4	0.2500
E	0.2500
F	0.2570
G	0.2610
17/64	0.2656
H	0.2660

(Continued)

Drill Size	Decimal Diameter
I	0.2720
J	0.2770
K	0.2810
9/32	0.2813
L	0.2900
M	0.2950
19/64	0.2969
N	0.3020
5/16	0.3125
O	0.3160
P	0.3230
21/64	0.3280
Q	0.3320
R	0.3390
11/32	0.3438
S	0.3480
T	0.3580
23/64	0.3590
U	0.3680
3/8	0.3750
V	0.3770
W	0.3860
25/64	0.3906
X	0.3970
Y	0.4040
13/32	0.4063
Z	0.4130
27/64	0.4219
7/16	0.4375
29/64	0.4531
15/32	0.4688
31/64	0.4844
1/2	0.5000
33/64	0.5156
17/32	0.5312
35/64	0.5469
9/16	0.5625
37/64	0.5781
19/32	0.5938
39/64	0.6094
5/8	0.6250
41/64	0.6406

Drill Size	Decimal Diameter
21/32	0.6562
43/64	0.6719
11/16	0.6875
45/64	0.7031
23/32	0.7188
47/64	0.7344
3/4	0.7500
49/64	0.7656
25/32	0.7812
51/64	0.7969
13/16	0.8125
53/64	0.8281
27/32	0.8438
55/64	0.8594
7/8	0.8750
57/64	0.8906
29/32	0.9062
59/64	0.9219
15/16	0.9375
61/64	0.9531
31/32	0.9688
63/64	0.9844
1	1.0000
1 3/64	1.0469
1 7/64	1.1094
1 1/8	1.1250
1 11/64	1.1719
1 7/32	1.2188
1 1/4	1.2500
1 19/64	1.2969
1 11/32	1.3438
1 3/8	1.3750
1 27/64	1.4219
1 1/2	1.5000

Appendix B1

Current Symbols

The majority of the current dimensioning and tolerancing symbols are the same as those used in the previous standards. A few new symbols have been added to more closely match international practices and to reduce the number of words placed on drawings.

Dimensioning Symbols		
Current practice	Abbreviation in notes	Parameter
	DIA	Diameter
	SPHER DIA	Spherical diameter
	R	Radius
	CR	Controlled radius
	SR	Spherical radius
	CBORE	Counterbore
	SF or SFACE	Spotface
	CSK -	Countersink
	DP	Deep
	—	Dimension origin
	SQ	Square
	REF	Reference
	PL	Places, times
	—	Arc length
	—	Slope
	—	Conical taper
	—	Basic dimension
	—	Statistical tolerance
	—	Continuous feature
	—	Between
	—	All around
	—	All over
	—	Datum feature

Dimensioning Symbols		
Current practice	Abbreviation in notes	Parameter
	—	Datum target
	—	Movable datum target
	—	Translation
	RFS	Regardless of feature size
	MMC/MMB	Maximum material condition Maximum material boundary
	LMC/LMB	Least material condition Least material boundary
	—	Free state
	—	Tangent plane
	—	Unequally disposed profile
	—	Independency
	—	Projected tolerance zone
	—	Straightness
	—	Flatness
	—	Circularity
	—	Cylindricity
	—	Perpendicularity
	—	Parallelism
	—	Angularity
	—	Position
	—	Symmetry
	—	Concentricity
	—	Circular runout
	—	Total runout
	—	Line profile
	—	Surface profile

Appendix B2

Symbols History

Standard symbols have changed slightly as the national dimensioning and tolerancing standard was revised. Symbols defined in previous standards are shown. Optional symbols contained in the 1982 standard are highlighted.

Symbols in Editions of Standards				
Current practice	1994	1982	1973	Parameter
\varnothing	\varnothing	\varnothing	DIA	Diameter
S \varnothing	S \varnothing	S \varnothing	SPHER DIA	Spherical diameter
R	R	R	R	Radius (redefined 1994)
CR	CR	none	none	Controlled radius
SR	SR	SR	SR	Spherical radius
			CBORE	Counterbore
			SF or SFACE	Spotface
			CSK	Countersink
			DP	Deep
			none	Dimension origin
			SQ	Square
()	()	()	REF	Reference
X	X	X	PL	Places, times
			none	Arc length
			none	Slope
			none	Conical taper
				Basic dimension
		none	none	Statistical tolerance
	none	none	none	Continuous feature
		none	none	Between
		none	none	All around
	none	none	none	All over

(Continued)

Symbols in Editions of Standards (continued)				
Current practice	1994	1982	1973	Parameter
				Datum feature
				Datum target
	none	none	none	Movable datum target
	none	none	none	Translation
none	none			Regardless of feature size
				Maximum material condition
	none	none	none	Maximum material boundary
				Least material condition
	none	none	none	Least material boundary
		none	none	Free state
		none	none	Tangent plane
	none	none	none	Unequally disposed
	none	none	none	Independency
				Projected tolerance zone
—	—	—	—	Straightness
				Flatness
				Circularity
				Cylindricity
				Perpendicularity
				Parallelism
				Angularity
				Position
				Symmetry
				Concentricity
				Circular runout
				Total runout
				Line profile
				Surface profile

Glossary

A

actual mating envelope: A true geometric counterpart of the feature that is at the maximum size that will fit within the hole or other internal feature or at the minimum size that will encompass a shaft or other external feature. (5, 7, 9)

actual mating envelope axis: The axis of an actual mating envelope. (7)

additional tolerance: Any increase in allowable tolerance that is permitted by the departure of a feature from the material condition specified in the feature control frame. Also called *bonus tolerance*. (8)

aligned dimensioning: A system of dimensioning in which dimension values are aligned with the dimension lines. Values are oriented to read from the bottom or right side of the page. This system is seldom used and is no longer illustrated in the dimensioning and tolerancing standard. (3)

all around symbol: A circle drawn at the corner in a leader that extends from a feature control frame. The all around symbol indicates that the tolerance applies all the way around the profile of the feature to which the leader is attached. (2)

all over symbol: Two concentric circles drawn at the corner in a leader that extends from a feature control frame. The all over symbol indicates that the tolerance applies all over the part profile. (2)

allowance: The difference between the maximum shaft size limit and the minimum hole size limit. (4)

Anglo-Saxon Foot: A distance standard previously used in Great Britain. The Anglo-Saxon Foot was based on the 11.65" long Roman Foot. (1)

angularity symbol: Two lines drawn to form a 30° angle, used for specification of an angularity tolerance. (2)

angularity tolerance: The amount of permitted error on the angular relationship between features. This tolerance is traditionally specified on angles other than 90° or parallel. (7)

ANSI: The abbreviation for American National Standards Institute. (1)

apex offset, cone: The distance from the apex of a cone to a line extending perpendicular to and from the center of the cone base. (4)

arc: Any segment of a circle other than a full circle. (4)

arc length: The circumferential distance measured along an arc. (2)

arc length symbol: An arc drawn above a dimension value, used to indicate that the given dimension defines an arc length. (2)

arc tangent: The point where an arc becomes tangent to another arc or a line. (4)

arrowhead: A symbol placed at each end of the dimension lines and on the end of a leader to indicate the point of application. (3)

as-designed: The features in a CAD model in their modeled condition. The as-designed condition is typically the design intent and the most desired condition for the produced part. The as-designed condition is not, according to ASME Y14.5-2009, a required manufacturing target. The drawing notes or referenced documents must establish it as a target when a specific target is required. (3, 4)

ASME: The abbreviation for the American Society of Mechanical Engineers. (1, 3)

axes: The three mutually perpendicular vectors that make up a coordinate system. Axes are typically labeled with capital letters. Degrees of freedom along the axes are typically labeled with lowercase letters. (2, 6)

axis method: A method where the axis (or center plane) of the actual mating envelope for a produced feature of size is used to determine if the axis is within the allowable position or orientation tolerance zone. (8)

B

base diameter, cone: The distance across the bottom of a cone. (4)

baseline dimensioning: A system in which all dimensions in one direction originate from the same feature. (3)

basic dimension: A theoretically exact value applied to the size or location of a feature. Title block tolerances do not apply. Tolerance must be applied by feature control frames or other means. (2, 3)

basic hole system: A system for calculating the limits of size for a mating hole and shaft. The size limits for the hole are calculated first, and the size limits for the shaft are based on the hole dimensions. (4)

basic shaft system: A system for calculating the limits of size for a mating hole and shaft. The size limits for the shaft are calculated first, and the size limits for the hole are based on the shaft dimensions. (4)

bidirectional position tolerance: The application of two position tolerances that allow differing amounts of variation in two directions on a single feature. (8)

bilateral profile tolerance: A profile tolerance that extends in both directions relative to the true profile of the feature. (11)

bilateral tolerance: An applicable amount of tolerance in both a plus and minus direction. (4)

blind hole: A hole that does not go all the way through the material in which it exists. (4)

bonus tolerance: Any increase in allowable tolerance that is permitted by the departure of a feature from the material condition specified in the feature control frame. (5, 8)

boss: A local protrusion on a part that is produced to provide a small area to machine. This prevents the need to machine an entire surface when a small locally machined area is adequate. (11)

boundary method: Synonymous with *surface method*. A method where the surface of a produced feature of size is used to determine if that surface is within the allowable boundary created by a specified position or orientation tolerance. (8)

British Imperial Yard: The permanent standard established in Great Britain in 1855. (1)

C

CAD: The abbreviation for computer-aided design. (1)

caliper: A tool used to make measurements. (5)

centerdrill: A tool used to produce a combined drilled hole and 60° countersink that is used for a machine center. (4)

center line: A drawing entity made up of alternating long and short dashes. It represents the center or axis of a part feature. (5)

chain dimensioning: A system in which dimensions go point-to-point to locate each consecutive feature from the previous one. (3)

chamfer: A small inclined surface cut on the edge of a part. A chamfer may be required for the purpose of removing sharp edges, and is commonly used to make assembly of parts easier. (4)

circular runout: Surface variation measured relative to an axis of rotation at a circular cross section. (10)

circular runout symbol: A single arrow with a filled or unfilled arrowhead, used for specification of a circular runout tolerance. (2)

circularity symbol: A circle symbol used for specification of a circularity tolerance. (2)

circularity tolerance: A form tolerance used to control the circular cross section of features such as cylinders and cones. (5)

clearance fit: A condition in which an internal part is smaller than the mating external part. (3)

clearance hole: A hole that has an MMC larger than the MMC of the fastener that goes into the hole. (8, 12)

clocking: Control of the rotational position of one or more features relative to another feature on a part or in an assembly. (6)

CMM: The abbreviation for coordinate measuring machine. See *coordinate measuring machine*. (5)

coaxial hole pattern: Two or more holes that lie on the same axis. An example is the set of coaxial holes for a hinge pin. (9)

coaxiality: The condition of two or more features being centered on the same centerline. (9)

composite position tolerance: A position tolerance specification that provides a pattern-locating tolerance and a feature-relating tolerance. (9)

compound datum feature reference: A term sometimes applied to a multiple datum feature reference. Two or more datum letters separated by a dash and placed in one cell of a feature control frame or shown in a note. Datum feature reference B-C indicates that features B and C are to be used to establish one datum. (6)

compound datum features: A term sometimes applied to multiple datum features. Two or more datum features that act together to establish a single datum plane or axis. (6)

concentricity: A theoretically perfect relationship where the derived median line of one feature coincides with the axis of another feature. This is a tolerancing definition and not intended to serve as a mathematical definition. (9)

concentricity symbol: A symbol consisting of two concentric circles, used for specification of a concentricity tolerance. (2)

concentricity tolerance: An allowable variation of the location of a derived median line for one feature relative to the axis of another feature. This is a very difficult tolerance to verify. Runout and position tolerances are normally preferable. (9)

cone angle: The angular measurement across cone elements in a right circular cone with the apex acting as the vertex of the angle. (4)

cone height: The distance from the base to the apex of a cone. (4)

continuous feature: A single feature with interruptions that are to be disregarded when considering the requirements for the feature. This is sometimes described as multiple features acting as one continuous feature. (6)

continuous feature (CF) symbol: An elongated hexagon containing the letters CF, used to indicate that a dimension or tolerance applies across interruptions in a feature of size so that it acts as a single continuous feature. (2)

controlled radius: A specification used to describe an arc without any reversals. (4)

controlled radius symbol: The abbreviation CR placed as a prefix to a dimension value, used for a controlled radius specification. (2)

coordinate measuring machine: A machine that electronically records coordinates for features on a part. A coordinate measuring machine may also calculate feature variations from specified dimensions and report any existing discrepancies. (5)

coplanarity: Two or more features in the same plane. (11)

corner radius: A small radial feature produced by machining. Machine cutters typically include a small radius at the end of the cutter, which produces a corner radius at locations such as the bottom of a counterbore. Corner radii are typically desired to reduce stress concentrations. (2, 4)

counterbore: A stepped increase in the diameter of a hole. Counterbores are located at the ends of holes. (4)

counterbore symbol: A symbol shaped like the bottom of a counterbore, placed before the diameter symbol and the numerical value representing the diameter of the counterbore. (2)

counterdrill depth: The distance from the entrance of the hole to the bottom of the large diameter. (4)

counterdrilled hole: A stepped hole made of two diameters located on the same centerline. The transition between the two diameters is conical and is formed by the drill angle on the large diameter drill. (4)

countersink: An angular increase in the size of a hole. A countersink has the shape of a right circular cone and is coaxial with the hole. (4)

countersink angle: The cone angle of the countersink. Countersink angles for screw heads must match the screw head angle. (4)

countersink symbol: A V-shaped symbol that has the appearance of a countersink, used to indicate a countersink dimension. (2)

cover sheet: The top sheet of a separate notes or parts list that includes general information such as the drawing number and title. (3)

cubit: An ancient distance standard, equivalent in length to the Egyptian pharaoh's forearm. (1)

cylinder: A geometric shape formed by revolving a line around an axis that is parallel to the line. This shape has a diameter and a length. (4)

cylindricity symbol: A circle with two tangent lines, used for specification of a cylindricity tolerance. (2)

cylindricity tolerance: The allowable amount of variation from the shape of a perfect cylinder. The tolerance boundary is formed by two perfect cylinders. (5)

D

datum: Theoretical point, line, or plane established from datum features on a part. (6)

datum axis: An axis established at the center of a true geometric counterpart for a datum feature such as a hole or shaft. (6)

datum feature: Physical feature on a part that is used to establish the location of a theoretically perfect datum. (2, 6)

datum feature of size: A physical feature that has size, such as a hole, that is identified with a datum feature symbol. (6)

datum feature reference: Placement of a datum letter in a feature control frame or in a note that specifies requirements relative to datums. (6)

datum feature symbol: A square or rectangle drawn around a letter with a leader extending to a triangle. This symbol is used to identify datum features on a drawing. (2, 6)

datum feature triangle: A triangle symbol, typically shown filled, attached to a leader line extending from the square of a datum feature symbol. (2)

datum precedence: The order in which datums are referenced in the tolerances and the order in which they are established when fabricating or inspecting a product. (6)

datum reference frame (DRF): Three planes that establish a coordinate system. A datum reference frame is created by datum feature references in a feature control frame or note. One part may have multiple datum reference frames. (2, 6)

datum simulation: The use of a tool to contact a datum feature and establish the location of the part relative to the datum. (6)

datum simulator: A tool used to contact a datum feature. The tool probably contains some error and therefore does not establish a perfect datum, but adequately simulates the datum for practical purposes. (6)

datum target: Point, line, or area on a feature that is identified to establish a datum. (2)

datum target area: An area on a surface that is contacted to establish a datum. (2)

datum target line: A line along a surface that is contacted to establish a datum. (2)

datum target point: A point on a surface that is contacted to establish a datum. (2)

datum target symbol: A circle containing a letter and number to identify datum targets and associate them with a specific datum. (2, 6)

decimal inches: Inch values with decimal divisions. (3)

decimal places: The number of digits to the right of the decimal point. (3)

degrees of freedom: The allowed movement of a part. There are three translational degrees of freedom along the X, Y, and Z axes. There are three rotational degrees of freedom around the axes. Rotations are labeled as *u* around the X axis, *v* around the Y axis, and *w* around the Z axis. Degrees of freedom notations are made with lowercase letters. (6)

departure from MMC: The amount of size variation from the maximum material condition of the feature. (5)

depth symbol: A downward-pointing arrow extending from a horizontal line and placed in front of a dimension value, used for the specification of depth for a feature. (2)

derived median line: A line that may not be straight (curve) that is made of the midpoints between two surface elements that are on opposite sides of the feature or by center points at all cross sections along the length of a cylindrical shape. The cross sections are normal (perpendicular) to the axis of the unrelated actual mating envelope. (5, 7, 9)

derived median plane: An imperfect (abstract) plane formed at the midpoints of all lines extending between the opposite faces of parallel surfaces. The lines are normal (perpendicular) to the center plane of the unrelated actual mating envelope. (9)

dial indicator: A device used to measure variations relative to a desired condition. A dial indicator does not measure size. (5)

diameter: The distance across a circle measured on a line that passes through the center. (4)

diameter symbol: A circle with a diagonal line through it. A diameter symbol is placed in front of numbers to indicate the value is a diameter. (2)

digit: A distance standard used 5000 years ago. A digit is based on the distance across a finger. (1)

dimension line: A line used to indicate the direction of the dimension. Dimension lines are drawn parallel to the dimensioned feature. (3)

dimension origin: The end of a dimension that serves as the starting point. (3)

double dimensioning: A condition in which the size, location, or tolerance of a feature is specified more than once. This is not allowed by the dimensioning and tolerancing standard. (3)

drill depth: The distance from the penetrated surface to the bottom of the full diameter. (4)

E

envelope: A perfect geometric shape corresponding to the design shape. It may be a fixed size such as the perfect form boundary or it may be a variable size that expands or contracts to make contact with a produced and imperfect feature. (6)

envelope principle: A requirement specifying that a boundary of perfect form exists at the MMC of any feature of size unless otherwise specified. See Rule #1. (5)

equal bilateral profile tolerance: A profile tolerance that extends equally in both directions relative to the true profile of the feature. Also called *equally disposed profile tolerance*. (11)

equalizing datum targets: Targets located such that when they are contacted, the forces on the part result in a centering effect. (6)

extension line: A line used to extend dimensioned features to the dimension lines. Extension lines make it possible to define part size without placing dimensions on the view. (3)

extrusion: A shape formed by pushing or pulling material through a die. (4)

F

feature control frame: A tolerance specification in a format that shows the tolerance type, tolerance value, any applicable material condition modifiers, and any applicable datum feature references. (2, 3, 5)

feature of size: Generally, a single feature or group of features defining a single shape that affects the weight of a part if its dimension is changed. As an example, changing a hole diameter affects how much material is in the part, thereby affecting the weight of the part. There are two classes for features of size: regular and irregular. (4)

feature of size, regular: Typically one part feature that is associated with a single size dimension such as thickness, width, length, or diameter. (4)

feature of size, irregular: An enclosed shape or a collection of features that in combination establish a boundary. The shape may be a closed boundary such as a D-shaped hole that has a profile tolerance applied, or it may be a shape such as a cylinder that is inscribed within and tangent to three pins. (4)

feature-relating tolerance (position or profile): The tolerance value in the second line of a composite position tolerance. This tolerance controls locations between features in a pattern. (9)

feature-relating tolerance zone framework (FRTZF): The relative true positions of the features in a pattern. (9)

first angle projection: An orthographic view creation system where visible features are determined by placing a viewing plane on the side opposite the adjacent view in relationship to the viewing position. (1)

fixed fastener condition: An assembly condition in which two or more parts are stacked together with a fastener passed through them. One of the parts has a hole that in some manner fixes the location of the fastener. The hole may be threaded with the fastener threaded into it, or the hole may be sized for a press fit. (8)

flatness symbol: A parallelogram symbol used for specification of a flatness tolerance. (2)

flatness tolerance: A tolerance control that defines the allowable amount of variation relative to a perfect plane. (5)

floating fastener condition: An assembly condition in which two or more parts are stacked together with a fastener passing through them. All of the parts have clearance holes in them. (8)

force fit: A combination of sizes on a shaft and hole that results in an interference fit. The shaft is larger than the hole. (4)

foreshortened radius: A radius seen in a view where the line of sight is not perpendicular to the plane in which the arc is located. (4)

form: The condition of a single feature such as a straight rod, flat surface, round hole, or cylinder. (5)

form tolerance: The amount of allowable variation of a single feature from the theoretically perfect form of that feature. (5)

free state symbol: The letter F inside a circle, used for specification of tolerances intended to apply in the free state when other tolerances on the same feature are specified to apply in a restrained condition. (2)

free state variation: The condition of a part that permits its dimensions to change after the part is removed from the manufacturing equipment. (5)

full indicator movement (FIM): The difference between the minimum and maximum reading obtained with a dial indicator. Also referred to as *total indicator reading (TIR)*. (5)

functional gage: A tool that can be used to check dimension limits by placing the gage on or in the workpiece. An example is a plate with four pins that check the locations of four holes. (5)

G

general note: A note that contains information that applies to an entire drawing. General notes are precise in specifying requirements. (3)

geometric dimensioning and tolerancing (GD&T): A general term used for tolerances applied to part features and most often used in reference to tolerances of form, orientation, location, profile, and runout. (1)

H

hole identification symbol: A shape or letter used to identify holes of a common size and to relate those holes to a particular hole specification in a note or table. (3)

I

implied datum: The practice of specifying only the tolerance type and tolerance value, leaving the datum selection to the judgment of the manufacturer and inspector. This practice is now prohibited by the dimensioning and tolerancing standard. (8)

inch: A standard unit of measurement approximately 25.4 millimeters long. In 1324 AD, an inch was the distance equal to the length of three barley corns placed end-to-end. (1)

inclined surface: A surface at any angle other than parallel or perpendicular to another surface. (3)

independency symbol: The letter I inside a circle, used for identifying a feature of size that is not required to have perfect form at its maximum material condition. (2)

interference fit: A condition in which an external feature, such as a pin, is larger than the internal feature, such as a hole, into which the external part is assembled. (3)

irregular curve: For purposes of this text, any curve that does not have a constant radius. (4)

irregular feature of size: An enclosed shape or a collection of features that in combination establish a size boundary. (2)

J

joggle: An offset in an extension line that can be used to provide clearance between the extension line and other lines or symbols. (3, 4)

K

keyseat: A rectangular slot cut in a shaft or hub for the purpose of installing a key to lock rotational relationships between parts. (4)

L

lay: The direction of surface irregularities caused by machining processes. (4)

leader line: A line used to connect specific information to a point on a drawing. Also called *leader*. (3)

least material boundary (LMB): The limit within the material that a feature may extend to as a result of size and geometric tolerances that are applicable to the feature. Surfaces and features of size have material boundaries. (2, 6)

least material condition (LMC): The condition of a feature when at the size limit that results in the least material in the part. Maximum hole diameter and minimum shaft diameter are examples of the least material condition. Reference to LMC encompasses LMS. (3, 5, 6)

least material size (LMS): The size limit of the feature at which the least material is allowed. The least material size is the size of the least material condition. (6)

limit dimension: A dimension that shows the minimum and maximum allowable values for a feature size or location. (4)

line fit: A calculated set of limits for two parts that results in a zero allowance. (3)

line profile: Control of line elements in one direction on a surface. (11)

LMB: The abbreviation for least material boundary. See *least material boundary*. (6)

LMC: The abbreviation for least material condition. See *least material condition*. (3, 5, 6)

location clearance fit: A condition in which there is clearance between mating parts. The amount of clearance is based on the desired accuracy of location. This type of fit is not used for rotating parts. (4)

location dimension: A value that indicates where one feature belongs relative to another. (3)

location interference fit: A condition in which the shaft is larger than the hole. The amount of interference is only meant to provide a known relative position for the parts. The interference is not meant to hold the two parts together. (4)

location tolerance: The specified allowable variation on where a feature is supposed to be in relationship to other features. (2)

location transition fit: A condition that may result in either a slight interference or clearance. This fit is used to provide a good relative location between the two parts. (4)

location variation: The distance a produced feature varies relative to the specified basic or as-designed location. (8)

M

machine relief: A feature produced in a part to provide machine cutter clearance. Also called a *machining relief*. (4)

mate drill: An operation in which two parts are placed together and both parts drilled in one operation. (4)

mated assemblies: Assemblies with parts that are mate drilled or mate machined and are not interchangeable with parts from other assemblies. (4)

material boundary modifier: A symbol used on datum feature references to indicate at what material boundary the specified datum feature is applicable. (3)

material condition modifier: A symbol used on tolerance values in feature control frames to indicate at what material condition the specified tolerance value is applicable. (3)

maximum material boundary (MMB): The limit outside the material that a feature may extend to as a result of size and geometric tolerances that are applicable to the feature. Surfaces and features of size have material boundaries. (2, 6)

maximum material condition (MMC): The condition of a feature when at the size limit that results in the maximum material in the part. Minimum hole diameter and maximum shaft diameter are examples of the maximum material condition. Reference to MMC encompasses MMS. (3, 5)

maximum material size (MMS): The size limit of the feature at which the maximum material is allowed; the size of the maximum material condition. (5, 6)

meter, standard: A standard established in 1798 based on one ten-millionth of the distance from the north pole to the equator when measured on a line passing through Paris, France. This standard has been replaced by the US Prototype Meter number 27. (1)

micrometer: A tool used for making measurements. (5)

MIL-STD-8: A series of military standards issued starting in the 1950s to specify dimensioning and tolerancing methods. The current dimensioning and tolerancing standard evolved from this series. (1)

minor radius, countersink: The shortest distance between the axis of a hole and the intersection of the countersink with an irregular curved surface. (4)

MMB: The abbreviation for maximum material boundary. See *maximum material boundary*. (6)

MMC: The abbreviation for maximum material condition. See *maximum material condition*. (3, 5)

modifier: A symbol placed following a tolerance value or datum feature reference that modifies the meaning or applicability. (2)

movable datum target: A datum target that is required to move to make contact with the datum feature. (2)

movable datum target symbol: A datum target symbol that includes two tangent lines forming a 60° angle and centered on a horizontal line through the symbol, used to identify a datum target and indicate the datum target simulator is movable. (6)

multiple datum feature reference: Two or more datum letters separated by a dash and placed in one cell of a feature control frame or shown in a note. Datum feature reference B-C indicates that features B and C are to be used to establish one datum. (10)

multiple datum features: Two or more datum features that act together to establish a single datum plane or axis. Also called *compound datum features*. (10)

multiview drawing: An orthographic drawing that shows the required views to describe a part. (3)

N

nominal size: The size or condition of a feature by which a feature is known, and which may not be the as-designed value or either dimension limit for the feature. An example is a 1/4" pipe, which is not .250" diameter. Nominal is a term often misused to indicate the as-designed condition. (4)

normal: At an angle generally thought of as perpendicular. In the case of an ellipse, a line extending from a point on the ellipse is normal to the ellipse if the line is perpendicular to a tangent line that passes through the point. (10)

note flag: A symbol and number placed on a drawing to indicate that a note in the notes list is applicable. Note flags are often attached to specific features with a leader. (3)

note sheets: Usually pages of a separate notes and parts list that contain the drawing notes, but also a sheet or sheets within a drawing. (3)

O

Olympic Foot: The official name for the 16 digit foot established by the Greeks. Its length is 12.16". (1)

order of precedence: The order in which datum feature references are made to indicate which datum is primary, secondary, and tertiary. (6)

orientation: The angular relationship of one feature to other features. (7)

orientation tolerance: An allowable variation expressed in a perpendicularity, parallelism, or angularity tolerance. (7)

orientation variation: The angular variation of a produced feature relative to the basic or as-designed orientation. (7)

origin of measurement: The point, line, or plane from which measurements are made. (2)

origin symbol: A symbol placed on a feature or extension line to indicate where a dimension starts. (2)

orthographic projection: A projection system where the object is rotated 90° between adjacent views with the axis of rotation perpendicular to the lines of projection between the views. (1)

P

palm: The distance across the palm of a hand. The palm was used as a distance standard 5000 years ago. (1)

paper gaging: A method of analyzing collected inspection data to see if the part meets tolerance requirements. (8)

parallelepiped: A six-sided prism where all surface intersections form 90° angles. (4)

parallelism symbol: A symbol consisting of two inclined parallel lines, used for specification of a parallelism tolerance. (2)

parallelism tolerance: The allowable orientation error of a feature relative to a parallel feature. This tolerance does not control location. (7)

partially fixed fastener: A condition where a fastener passes through clearance holes, but the position tolerance on at least one of the holes is larger than the amount of clearance that exists when the hole is at MMC. (12)

parts list sheets: One or more pages that list the parts in an assembly or the material for a detail part. (3)

pattern-locating tolerance (position or profile): The tolerance value in the first segment of a composite position tolerance. This tolerance controls locations and orientation of the features relative to the referenced datums. (9)

pattern-locating tolerance zone framework (PLTZF): The true positions of the controlled features relative to the referenced datums. (9)

perpendicularity symbol: A symbol consisting of two perpendicular lines, used for specification of a perpendicularity tolerance. (2)

perpendicularity tolerance: The amount of acceptable variation on a 90° angle between a feature and a referenced datum. (7)

pitch diameter: The diameter of the pitch cylinder created by threads. The pitch diameter is approximately midway between the root and crest of the threads. (2)

plus and minus tolerance: Values that show the allowable variation above and below the specified dimension value. It is possible for one of the tolerance values to be zero. (4)

polar coordinate: A point defined by a combination of an angle dimension and a distance from an origin point. (3)

polar coordinate dimensioning system: The application of dimensions from a common center point and an angle. (3)

position tolerance: A value that defines the acceptable amount of variation in the location of a feature relative to the true position specified for the feature. (8)

position tolerance symbol: A symbol consisting of two lines crossing at the center of a circle, used for specification of a position tolerance. (2)

position tolerance zone: The acceptable zone created by a position tolerance. It may be the allowable space for an axis or center plane for a feature of size, or it may be a boundary that must not be violated by the surface of a feature with a position tolerance. (8)

positional tolerancing: A term sometimes used in reference to geometric dimensioning and tolerancing. (1)

primary datum: The first datum feature reference in a feature control frame. If the primary datum is a surface, at least three points on this surface must be contacted to establish the primary plane. (6)

profile of a line: A control of line elements in only one direction on a surface. (11)

profile of a line symbol: A semicircle symbol used for specification of a line profile tolerance. (2)

profile of a surface: A simultaneous control of all points over the full area of a surface. (11)

profile of a surface symbol: A semicircle with a horizontal line drawn across the bottom, used for specification of a surface profile tolerance. (2)

profile tolerance: A tolerance on a feature that can specify any needed control ranging from a singular control of form to a simultaneous control of form, orientation, and location. (11)

projected tolerance zone: A tolerance zone that extends to the outside of the controlled feature. (8, 12)

projected tolerance zone symbol: The letter P within a circle, used for identification of a tolerance zone that is required to project outside the object. (2)

projection lines: Lines either shown or imagined that extend between the same feature in adjacent views. (1)

R

radial distance: A distance measured along a line extending from the center of a feature such as a cylinder to any point on the feature. (5)

radial hole pattern: A pattern of holes located on radial lines. (9)

radial line: A line extending from the center of a round feature such as a cylinder. (5)

radius symbol: The letter R placed in front of the radius dimension value. (2)

rectangular coordinate dimensioning system: The application of dimensions that originate at three mutually perpendicular planes. (3)

rectangular coordinate dimensioning without dimension lines: A system of dimensioning in which the dimension values are shown at the ends of the extension lines. No dimension lines are shown. (3)

reference dimension: A dimension that provides information about location or size but does not indicate a requirement. A dimension value is enclosed in parentheses to indicate that the value is for reference only. (3)

reference value: A number shown on the drawing for information purposes. A reference value does not establish a firm requirement for the acceptance or rejection of parts. (2)

regardless of feature size (RFS): The default condition for an applied tolerance value that requires the tolerance to be met regardless of the feature size. (3)

regardless of material boundary (RMB): A default condition for datum feature references that requires the datum feature to be established from the feature surface regardless of its material condition. (2)

regular feature of size: Typically one part feature that is associated with a single size dimension such as thickness, width, length, or diameter. (2, 4)

related actual mating envelope: An actual mating envelope that is related to one or more datums as required for the geometric tolerances that are applied to the feature. (6, 7)

removed view: A view that is placed away from its normal location. It may be an orthographic view located out of its normal location or a small segment of the part shown out of its projected position. (4)

revisions: Changes made after the initial release of a drawing or model. (4)

right square pyramid: A pyramid with its apex located on a line that is perpendicular to and centered on the base. (4)

Roman Foot: A Roman unit of measure equal to about 11.65". (1)

rotational constraint: The limitation of angular movement about an axis of the datum reference frame. (6)

roughness: The small peaks and valleys on a surface that are caused by manufacturing processes. (4)

roughness cutoff: The distance across which roughness values are measured. Also referred to as *roughness width cutoff* or *sampling length*. (4)

roughness width: The distance across peaks and valleys that cause surface roughness. (4)

round off: Creating a value with a predetermined number of decimal places from a value that has a greater number of decimal places. (3)

round off error: The error in answers obtained when calculating with rounded off numbers. (3)

Royal Cubit: A permanent distance standard made of black granite by the Egyptians and based on the cubit length (20.67"). The Royal Cubit was used to calibrate measuring sticks. (1)

Rule #1: A rule in the dimensioning and tolerancing standard stating that any dimensioned feature of size must have perfect form when the feature is produced at its maximum material condition. (5)

Rule #2: A rule in the dimensioning and tolerancing standard stating that all tolerances are assumed to apply regardless of feature size and datum feature references apply regardless of material boundary unless a modifier is shown to indicate otherwise. (4)

running and sliding clearance fit: A condition in which the shaft is always smaller than the hole and the parts are free to move. (4)

runout: Variation of a surface relative to an axis of rotation. (10)

S

sampling length: The distance across which roughness values are measured. Also referred to as *roughness cutoff*. (4)

scale: The proportion of the drawing relative to the actual size of the part. (3)

secondary datum: The second datum feature reference in a feature control frame. This datum must be established after the primary datum. (6)

sharp diameter, flathead screw: The diameter created by extending the screw head cone all the way to the top of the head. (4)

SI (metric) system: A measurement system in which values are based on the millimeter. (3)

simultaneous requirement: A condition where two or more features act as one pattern as a result of two or more feature control frames referencing the exact same datum reference frame. (9)

sine bar: A device that has a well-controlled distance between mounting feet, and is used to set up angles. The length is fixed at a distance that permits stacking blocks under one foot to produce a height equal to a multiple of the sine of the angle to be established. (5)

single limit dimension: A dimension that only shows one acceptable limit. The other limit must be controlled by the geometry of the part. (4)

sixteen digit foot: A distance standard established by the Greeks (12.16"). A sixteen digit foot was approximately two-thirds the length of a Greek Cubit. (1)

size dimension: A value that defines how large a feature is. (3)

slope: A ratio of the rise over the run of an inclined surface. (2)

slope symbol: A triangle symbol placed in front of a dimension value, used in a slope specification for a flat surface. (2)

span: A distance standard used 5000 years ago. The span was based on the distance from the end of the thumb to the end of the small finger when the hand is spread. (1)

specific note: Information attached to one or more specific features on a drawing. (3)

spherical diameter symbol: The letter S and diameter symbol placed in front of the spherical diameter dimension value. (2)

spherical radius: The distance from the surface of a spherical feature to its center point. (4)

spherical radius symbol: The letters SR placed in front of the spherical radius dimension value. (2)

spotface: A stepped increase in the diameter of a hole. A spotface is similar to a counterbore, but generally is not used to recess a screw head or other part. Spotfaces are generally used to provide good bearing surfaces for screw heads or to establish a controlled material thickness. (4)

spotface symbol: A symbol shaped like a spotface containing the letters SF, placed before the diameter symbol and the diameter value specifying the size of the flat created on the surface. (2)

square symbol: A small square placed in front of a dimension value to indicate that the dimension applies across both sets of flats on a square part. (2)

station point: Point located by tabulated coordinates. (3)

statistical tolerance symbol: An elongated hexagon containing the letters ST, used to identify a tolerance calculated by statistical methods. (2)

stepped datum surfaces: Two or more non-coplanar surfaces used to establish a single datum. (6)

stock shape: A geometric cross section of stock material that may commonly be purchased. Examples are round, square, rectangular, and hexagonal bar stock. (4)

straightness symbol: A short, straight horizontal line used for specification of a straightness tolerance. (2)

straightness tolerance: The acceptable amount of variation relative to a straight line. (5)

surface: Any face on a feature of size or surface feature. (4)

surface feature: Any flat or curved feature that is not defined by a size dimension. (11)

surface line element: A line on the surface of a part. It is usually created by intersecting a plane or cylinder with a surface on the part. (5)

surface method: Synonymous with *boundary method*. A method where the surface of a produced feature of size is used to determine if that surface is within the allowable boundary created by a specified position or orientation tolerance. (8)

surface profile: The simultaneous control of all points across an entire surface contour. (11)

symbol size: An established size for symbols proportional to the size of dimensioning characters used on a drawing. (2)

symmetrical features: A group of features that are equally located and shaped relative to a center plane or axis. (9)

symmetry: A condition where features on each side of a centerline or center plane are equal. (9)

symmetry symbol: A symbol consisting of three horizontal lines, used for specification of a symmetry tolerance. (2)

T

tabulated dimensions: Coordinate and size values contained in a table and referenced to the drawing by symbols or letters placed at the locations of the dimensioned feature. (3)

tangent plane: A plane that contacts the high points on a surface. (7)

tangent plane symbol: The letter T within a circle, used for identification of a tolerance that applies to a tangent plane instead of to a surface. (2)

taper: A ratio of the change in diameter per unit of length. (4)

taper symbol: A symbol that appears as a cone on an axis, used in a taper specification for a part. (2)

target area: A portion of a surface that is identified as a target. The target area is to be contacted with a tool that is sized to the dimensions of the target area. (6)

target line: A line on a surface that is to act as a target. The side of a dowel pin can be positioned to contact the line and establish the datum. (6)

target point: A single point on a surface that is to act as a target. A spherical tool located to contact the point can be used to establish the datum. (6)

tertiary datum: The third datum feature reference in a feature control frame or note. A tertiary datum is established after the primary and secondary datums are already established. (6)

third angle projection: An orthographic view creation system where visible features are determined by placing a viewing plane between the viewing position and the adjacent view. (1)

times/places: The number of occurrences of a feature; indicated by a number followed by an "X". (2)

title block tolerance: Allowable dimensional variations associated with untoleranced dimensions on the drawing. The tolerance values may be shown in the title block or notes list. (3)

tolerance: The acceptable dimensional variation that is permitted on a part. (1, 4)

tolerance accumulation: The addition of tolerances associated with features on a part. Also referred to as *tolerance buildup* and *tolerance stackup*. (3)

tolerance symbol: A geometric shape used in place of abbreviations for the communication of tolerances. (5)

tolerance value, feature control frame: The number following the tolerance symbol. This value indicates the permitted amount of variation. (5)

total indicator reading (TIR): The difference between the minimum and maximum reading obtained with a dial indicator. Also referred to as *full indicator movement (FIM)*. (5)

total runout: The amount of variation measured across an entire surface when the part is rotated on an axis of rotation. (10)

total runout symbol: Two arrows connected by a horizontal line with filled or unfilled arrowheads, used for specification of a total runout tolerance. (2)

transition fit: Limits of size applied to two parts that can result in either a clearance or interference fit. (3)

translation symbol: A symbol consisting of an equilateral triangle pointing to the right, used with a datum feature reference to indicate that the datum feature simulator may translate. (2)

translational constraint: The limitation of movement along an axis of the datum reference frame. (6)

true geometric counterpart (TGC): A perfect geometric shape, or a theoretically perfect simulator, corresponding to the shape of a datum feature on a part. (6)

true position: A theoretically exact location for a feature. A specified tolerance defines the allowable variation relative to the true position. (8)

true profile: The perfect as-designed geometry defined by basic dimensions. (11)

true radius: The actual distance between an arc center and the arc. This is different from the apparent radius seen when viewing an arc at any angle other than perpendicular to the plane of the arc. (4)

U

unequally disposed profile tolerance: A profile tolerance zone that is offset entirely or unequally relative to the true profile. (11)

unequally disposed tolerance symbol: The letter U inside a circle, used in a tolerance specification to indicate that a profile tolerance is unequally disposed. (2)

unidirectional dimensions: A system of dimensioning in which all numbers are written horizontally and read from the bottom of the page. (3)

unilateral profile tolerance: A profile tolerance that extends all of the tolerance zone in one direction relative to the true profile of the feature. It is specified using an unequally disposed symbol in a profile tolerance. (11)

unilateral tolerance: An applicable amount of tolerance in only one direction with zero tolerance in the other direction. (4)

unilateral zone: A tolerance zone that extends entirely in one direction. (2)

unrelated actual mating envelope: An actual mating envelope that is not related to any datums and that encompasses all size and form variations on the feature. (6, 7, 9)

US customary (inch) system: A measurement system in which values are based on the inch. (3)

US Prototype Meter number 27: The standard against which all US measurements are referenced. The length of this standard is established relative to a specific wavelength of light. (1)

V

virtual condition: The combined effect of the maximum material condition and any applicable tolerance. (5, 7)

W

waviness: Large surface variations on which roughness variations may be superimposed. (4)

workpiece: The part on which work is being performed. (5)

Y

yard: A distance standard originally established in the 12th century. This unit of measurement was equivalent to the distance from King Henry's nose to his thumb. A yard is now defined as 36". (1)

Z

zero allowance: The condition where the difference between the maximum shaft size limit and the minimum hole size limit is zero. (3)

zero tolerancing at MMC: A position or orientation tolerance applied at MMC or LMC that shows no tolerance in the feature control frame, but instead places all the tolerance on the size of the located feature. (12)

Index

A

abbreviations, 24–29, 70
actual mating envelope, 134, 187, 214, 283
actual mating envelope axis, 216
additional tolerance, 141–142, 254, 261 (*see* bonus tolerance)
 defined, 253
aligned dimensioning, 48
all around symbol, 32
all over symbol, 32
allowance, 105–106, 109
angles, 91–92
 tolerances on, 105
angles of projection, 18
Anglo-Saxon Foot, 10
angularity, 231–234
angularity symbol, 29
apex offset, 84
arc length symbol, 28
arc tangents, 94
arcs, 94–95
arrowheads, 47
ASME Y14.5, rules in, 110–113
assembly, 357–361
axes, 164–165 (*see* cylindrical features)
axis, 134

B

base diameter, 84
basic dimension, 36–38, 72–73
 noted, 36–37
basic dimension symbol, 36
basic hole system, 106
basic shaft system, 106–107
bidirectional position
 tolerance, 265
bilateral profile tolerance, 327–328
bilateral tolerance, 105
blind holes, 87
bonus tolerance, 141–142, 254, 261 (*see* additional tolerance)
 defined, 141, 253
 standards, 141
boundary method, 241

British Imperial Yard, 10
broken lengths, 103

C

calipers, 11
center line, 134
centerdrill, 101
centerdrilled holes, 99–101
chain dimensioning, 53
chamfers, 92
characters, size and style, 47–48
circular runout, 305–313
 applications, 306–309
 cylinder, 306–307
 datum axis, 309
 defined, 305
 face surfaces, 308–309
 feature control frames, 306
 limited tolerance zone
 application, 311
 multiple datum features, 309–311
 noncylindrical features, 307–308
 one datum feature, 309
circular runout symbol, 29
circularity, 152–153
 cone, 153
 cylindrical surface, 152–153
circularity symbol, 29
clearance fit, 71
clocking, 189
coaxial hole patterns, 292–294
coaxial tolerances, 294–295
composite position tolerance, 272–280
 gaging, 286–287
 verification, 278–280
compound datum feature
 reference, 192
compound datum features, 192, 309
computer numerically controlled (CNC) machines, 20
computer-aided design (CAD), 20
computer-aided manufacturing (CAM), 21

concentric diameters, 84
concentricity, 298–299
concentricity symbol, 30
cone height, 84
cones, 84, 145
continuous feature (CF)
 symbol, 30
controlled feature, 31
controlled radius symbol, 26
coordinate location tolerances, ambiguity, 259–260
coordinate measuring machine (CMM), 21, 144
counterbore, 88–89
counterbore symbol, 26
counterdrill, 90
countersink, 89–90
countersink symbol, 28
cover sheet, 70
cubit, 10
cylinders, 83–84
 flats on, 84
cylindrical features, 164–165, 177 (*see* axes)
cylindricity, 153–154
cylindricity symbol, 29

D

datum, 159–205
 advanced concepts, 199–205
 defined, 152
 equalizing targets to locate planes, 196–197
 features, 34
datum axis, 309
datum feature, 159–161
 application to surfaces, 35
 as surfaces, 170–177
 restrained, 205
datum feature references, 159–205
 customized, 200–201
 defined, 160
 feature control frame, 178–192
 LMB, MMB, RMB, 181–183
 number of, 180–181
 special applications, 192–199
datum feature symbol, 34–35, 163

- datum feature translation, 199–200
- datum feature triangle, 34
- datum features of size, 177–178
 - material boundary modifiers, 181–183
- datum identification, 34–36, 163–168
- datum precedence, 161
- datum reference frame (DRF), 36, 161–163
 - establishing from datum features, 168–170
 - related dimensions, 198–199
- datums,
 - establishing from datum features, 168–170
 - implied, 300
 - selecting for multiple parts, 358–359
 - special applications, 192–199
- datum simulation, 161, 169, 178
 - flat features, 174–175
- datum simulators, 161, 169
- datum symbols, 163–164
- datum target, movable, 35
- datum target areas, 35
- datum target lines, 35
- datum target points, 35
- datum targets, 35–36
 - based on order of precedence, 190–192
 - equalizing, 196
- datum target symbol, 35–36, 164
- decimal inches, 51
- degrees of freedom, 163
 - noting translational, 200
- depth symbol, 26
- derived median line, 134, 216, 297
- derived median plane, 297
- design, 14
- designers, 14
- dial caliper, 11
- diameter, 84
- diameter symbol, 25, 120
- digit, 10
- dimensional control,
 - development, 10–13
- dimension application, 82–103
 - angles, 91–92
 - arcs, 94–95
 - broken lengths, 103
 - centerdrilled holes, 99–101
 - chamfers, 92

- cones, 84
- counterbores, 88–89
- counterdrills, 90
- countersinks, 89–90
- cylinders, 83–84
- features on curved surfaces, 96
- flats on cylinders, 84
- foreshortened radii, 95–96
- holes, 84–87
 - for machine screws, 88–91
- irregular curves, 96–97
- joggles, 103
- keyseats, 98–99
- knurls, 101
- limits of size, 81–121
- machining reliefs, 101
- narrow spaces, 99
- past practices, 120–121
- prisms and pyramids, 82–83
- round-ended bars, 97–98
- sheet metal bends, 101–103
- slotted holes, 98
- spherical radii, 96
- spotfaces, 91
- standard sizes and shapes, 103
- symmetrical features, 97
- tapers, 92–94
- threads, 101
- dimension arrangement,
 - multiview drawings, 61–66
- dimension lines, 45–46
- dimension origin, 65
- dimension placement, multiview drawings, 59–61
- dimension revisions, 116–117
- dimension systems, 53–58
 - chain dimensioning, 53
 - direct (combined), 57–58
 - rectangular coordinate dimensioning, 53
- dimensioning,
 - according to related parts, 61
 - auxiliary view, 66
 - between views, 60
 - feature contour, 59–60
 - for industry, 113–117
 - hidden features, 64–65
 - line use, 45–47
 - off the views, 60–61
 - pictorial views, 68
 - requirements, 43–73
 - section view, 66–67

- tolerancing, 9–21
 - with notes, 68–70
- dimensioning methods, 44
- dimensioning skills,
 - application, 15–16
 - calculations, 17–18
 - discussion of principles, 17
 - interpretation, 16–17
 - required, 15–18
- dimensioning standards, 10–14
- dimensioning symbology, 23–38
- dimensions,
 - datum reference frame, 198–199
 - special views, 66–68
- direct (combined) dimensioning system, 57–58
- double dimensioning, 64
 - exception, 65
- drawing note, 10

E

- envelope, 198
- envelope principle, 131
- equalizing datum targets, 196
- extension lines, 45

F

- feature axis, 215–216
- feature center plane, 215–216
- feature control frame, 32, 73, 129, 212–214, 296–297
 - composition, 32–33
 - datum feature references, 178–192
 - features of size, 33
 - past format, 37–38
 - placement, 33–34
 - position, 242–244
 - surfaces, 33
 - threads, 34
- feature of size, 82
- feature-relating tolerance zone framework (FRTZF), 274
- feature-relating tolerance, 273, 368
- features of size, regular, 24, 82
- finish symbols, 120–121
- fit between parts, categories, 70–72
- fixed fastener condition, 252–253, 351–357
 - defined, 252
 - partially fixed fastener condition, 355–356

- three parts with unevenly distributed tolerances, 354–355
- three stacked parts, 353–354
- two parts with unevenly distributed tolerances, 354
- two stacked parts, 352–353
- flat surfaces, 164 (*see* planes)
- flatness, 147–153
 - applied to a surface, 147–150
 - defined, 147
 - derived median plane verification, 152
 - surface verification, 149–150
 - tolerance applied to derived median plane, 150–152
- flatness symbol, 29
- floating fastener condition, 251–252, 345–351
 - defined, 251
 - three stacked parts, 349–350
 - two parts with two hole sizes, 350–351
 - two parts with unevenly distributed tolerances, 350
 - two stacked parts, 346–349
- force fits, 108
- foreshortened radii, 95–96
- form tolerance, 127–154
 - applicable to single features, 130
 - categories, 128
 - combined with orientation, 235–236
 - feature control frames, 129–131
 - surfaces and features of size, 130
 - symbols, 128
- free state symbol, 31
- free state variation, 133
- full indicator movement (FIM), 138
- functional gages, 143, 257–258
- future trends, professions and trades, 20–21

G

- gaging, 281–284, 286–287
- geometric dimensioning and tolerancing (GD&T), 12
- geometric tolerances, 72–73

- geometric tolerancing symbols, 29–32 (*see* symbols)
- geometry, 117

H

- hardcopies, 20
- hole identification symbol, 56
- holes, 84–87
 - machine screws, 88–91
 - virtual condition, 142–143

I

- implied datums, 300
- inch measurement system, 51–52
- increased tolerance zone area, 260–261
- independency symbol, 32
- inspectors, 15
- interchangeability, 113–114
- interference fit, 71–72
- interpretation, 16–17
- interrupted feature, 31
- irregular curves, 96–97
- irregular features of size, 24, 82
- ISO standards, 44

J

- joggle, 55, 99, 103

K

- keyseat, 98–99
- knurls, 101

L

- lay, 117, 119–120
- leader lines, 46–47
- leaders, 46
- least material boundary (LMB), 31, 183
 - datum feature references, 183
 - datum feature surfaces, 203–204
 - datum features of size, 201–202
- least material condition (LMC), 31, 73, 131, 183
 - position tolerance, 248–249, 364
- least material size (LMS), 183
- limit calculations, 108–113
 - using tables, 108–109
 - without tables, 109–113
- limit dimensions, 104
- limits of application, 324

- limits of size, 103–105, 108
- line fit, 71
- line profile, 321–323
- lines, 167
- location dimension, 44–45
- location tolerances, 29

M

- machine capability, 114–116
- machine planners, 14
- machine programmers, 15
- machining reliefs, 101
- machinists, 15
- manufacturing engineers, 14
- mate drilling, 114
- material boundary modifiers, 73, 169, 214–215
 - datum features of size, 181–183
 - flat datum features, 175–177
 - offset datum features, 204–205
- material condition modifiers, 73, 129–130, 214–215
 - effect of, 246–250
 - feature control frame, 243–244
- maximum material boundary (MMB), 31, 169–170
 - datum feature references, 181–183
 - datum feature surfaces, 203–204
 - datum features of size, 201–202
 - primary datum feature reference, 185–186
 - secondary or tertiary datum feature reference, 187–189
 - surfaces, 201
- maximum material condition (MMC), 31, 73, 110, 131
 - perfect form, 121
 - position tolerance, 247–248
 - proof of concept, 249–250
 - virtual condition, 214–215
 - zero position tolerance, 362–364
- measured position variation diameter, 257
- measurement standards, 10–11
- measurement system, 10–11, 50–53
 - metric system, 52–53
- minor radius, 89

- modifiers, 30, 145, 183
- mold lines, 101
- movable datum target, 35
- movable datum target symbol, 196
- multiple (compound) datum features, 192–195, 309
- multiple feature control frames, same datum reference frame, 189–190
- multiple feature groups, one datum reference frame, 288–292
- multiple-segment tolerance specifications, 370

N

- narrow spaces, 99
- nominal size, 106, 322
- note sheets, 70
- notes, 68–70

O

- offset datum features, 204–205
- Olympic Foot, 10
- order of precedence, 171–174, 179–181
 - datum targets based on, 190–192
- orientation, 235–236
 - and form, 370–371
- orientation tolerance, 211–236
 - and form tolerance, 235–236
 - categories, 211
 - defined, 211
 - feature control frames, 212–214
 - material condition and material boundary modifiers, 214–215
 - surfaces and features of size, 213
 - symbols, 211–216
- origin symbol, 28
- orthographic projection, 18–20

P

- palm, 10
- paper gaging, 257, 364
 - techniques, 364–348
- paperless factory, 21
- parallel surfaces, 178 (*see* rectangular features)
- parallelepiped, 82

- parallelism, 216–220
 - cylindrical feature of size, 218–220
 - feature of size verification, 219–220
 - flat surface, 216–218
 - flat surface verification, 218
- parallelism symbol, 29
- partially fixed fastener, 346
- parts list sheets, 70
- pattern-locating tolerance, 273
- pattern-locating tolerance zone framework (PLTZF), 273
- perfect form boundary, 110, 131
- perpendicularity, 220–231
 - features of size, 225–231
 - flat surfaces, 221–224
- perpendicularity symbol, 29
- planes, 164, 172 (*see* flat surfaces)
- plus and minus tolerances, 104–105
 - avoiding on location dimensions, 58
 - location risk, 53
- polar coordinate dimensioning, 56–57
- position calculations, from coordinate measurements, 255–257
- position tolerance,
 - material condition modifiers, 246–250
 - multiple patterns, 287–295
 - no datum feature reference, 266–267
 - past practices, 299–300
 - regardless of feature size, 246–247
 - simultaneous patterns, 287–295
- position tolerance symbol, 30
- position tolerance zone, 244–246
 - basic dimensions and true positions, 245–246
 - datum reference frame, 246
- position tolerances, 242
 - advantages, 258–261
 - bidirectional position tolerance, 265
 - bonus tolerances, 253–255
 - bounded feature, 266–267
 - clarity, 258–260
 - composite, 272–280
 - counterbored holes, 262–263

- feature control frame, 242–244
- functional gaging, 280–287
- noncylindrical features of size, 265–266
- projected tolerance zones, 264
- special applications, 261–267
- threaded features, 263–264
- verifying, 255–258
- position tolerancing, 12
 - concentricity, 298–299
 - expanded principles, 271–300
 - symmetry, 295–298
- primary datum, 171–172
 - feature reference at MMB, 185–186
 - feature reference at RMB, 183–185
 - repeated, 276–277
- prisms, 82–83
- process, 119
- professions and trades, 14–15
 - designers, 14
 - future trends, 20–21
 - inspectors, 15
 - machine planners, 14
 - machine programmers, 15
 - machinists, 15
 - manufacturing engineers, 14
 - tool designers, 15
- profile, 321–340
 - all around application, 325–326
 - all over application, 326
 - application across the feature, 324–325
 - application between noted limits, 325
 - past practices, 340
- profile of a line symbol, 29
- profile of a surface symbol, 29
- profile specification, 321–330
 - past and alternative practices, 329–330
- profile tolerance zone
 - boundaries, 326–329
- profile tolerances, 321
- projected tolerance zone, 264, 356–357
- projected tolerance zone symbol, 31
- projection lines, 20
- pyramids, 82–83

R

- radial hole patterns, 295
- radius symbol, 26
- rectangular coordinate
 - dimensioning, 53–56
- rectangular features, 178 (*see* parallel surfaces)
- reference dimensions, 65
- referenced datums, 161
- regardless of feature size (RFS), 31, 73
 - position tolerance, 246–247
- regardless of material boundary (RMB), 31, 170
 - datum feature references, 181
 - datum feature surfaces, 203–204
 - datum features of size, 201–202
 - primary datum feature reference, 183–185
 - secondary or tertiary datum feature reference, 186–187
 - surfaces, 201
- regular features of size, 24, 82
- related actual mating envelope, 187, 202, 227
- removed view, 86
- restrained datum features, 205
- resultant condition, 248–249
- revisions, 113
- right square pyramid, 83
- Roman Foot, 10
- roughness, 117
- roughness cutoff, 118
- roughness width, 118
- roughness width cutoff, 118
- round-ended bars, 97–98
- rounding off numbers, 52
- Royal Cubit, 10
- Rule #1, 110–112, 131, 145
 - exceptions, 111–112, 133, 138
 - interrelationships, 112
- Rule #2, 112–113
- rule of simultaneity, 189
- runout, 305–317
 - referenced to two datums, 312–313
 - combined effects with size tolerances, 316–317

S

- sampling length, 118

- scale, 49–50
- secondary datum, 172–173
- separate requirements,
 - composite tolerances, 291–292
 - references to datum features of size, 289–291
- shaft, virtual condition, 140–141
- sharp diameter, 89
- sheet metal bends, 101–103
- SI (metric) system, 50
- simultaneity, rule of, 189
- simultaneous requirement, 189, 288
 - composite tolerances, 291–292
 - referenced to datum features of size, 288–289
- sine bar, 138
- single limit dimensions, 105
- sixteen digit foot, 10
- size dimension, 45
- size limits, 103–105
 - calculating with standard tables, 107–108
- slope symbol, 28
- slotted holes, 98
- span, 10
- specific notes, 68–69
- spherical diameter symbol, 26
- spherical radii, 96
- spherical radius symbol, 26
- spotface, 91–103
- spotface symbol, 26
- square symbol, 28
- staggered dimension values, 63
- standard sizes and shapes, 103
- station points, 56
- statistical tolerance symbol, 30
- stepped datum surfaces, 195–196
- straightness, 133–147
 - cone elements, 137
 - controlling a shaft derived median line, 140
 - cylinder surface elements, 136–137
 - derived median line between flat surfaces, 145–147
 - derived median line of a hole, 142
 - derived median line verification, 143–144
 - features of size, 139–144
 - flat surfaces, 136

- surface elements, 134–139
- surface verification, 137–138
- straightness symbol, 29
- surface line element, 134
- surface method, 241
- surface profile, 321, 323–324
- surface roughness, 117–118
- surface texture dimensions, 117–120
- surfaces, 82
 - as datum features, 170–177
- symbology, 23–38
- symbols, 24–29
 - application, 25–29
 - geometric tolerancing, 29–32
 - past practices, 37–38
 - shape and size, 29–30
- symmetrical features, 97
- symmetry, 295–298
 - features of size, 298
 - hole patterns, 297–298
- symmetry symbol, 30

T

- tangent plane, 234–235
- tangent plane symbol, 31
- tapers, 92–94
- taper symbol, 29
- target areas, 168, 197–198
- target lines, 167
- target points, 166, 197–198
- targets, 165–168
 - on diameters, 197
- teamwork, 14
- tertiary datum, 173–174
- theoretical datums, 161
- threads, 101
- title block tolerances, 52
- tolerance, 103
- tolerance accumulation, 53
- tolerance calculation, 250–255
- tolerance symbol, 129
- tolerance value, 129
- tolerancing,
 - derived median plane, 151
 - standards, 10–13
 - symbology, 23–38
- tool designers, 15
- total indicator reading (TIR), 138
- total runout, 313–317
 - applications, 314
 - cylinder, 314
 - defined, 313

- face surfaces, 314–316
- feature control frame, 314
- total runout symbol, 29
- translation symbol, 32
- true geometric counterpart (TGC), 161
- true position, 245
- true profile, 322
- two single-segment feature control frames, 280

U

- unequally disposed profile tolerance, 328, 330
- unequally disposed tolerance symbol, 31
- unidirectional dimensions, 48

- unilateral profile tolerance, 328–329
- unilateral tolerance, 105
- unilateral zone, 31
- unit area flatness tolerance, 150
- unit length tolerance, 144–145
 - verification, 145
- unrelated actual mating envelope, 187, 202, 215, 228, 283
- US customary (inch) system, 50
- US Prototype Meter number 27, 10

V

- views, 18–20
- virtual condition, 140, 214–215, 248–249

- hole, 142–143
- shaft, 140–141

W

- waviness, 117–119

Y

- yard, 10

Z

- zero allowance, 72
- zero tolerancing at MMC, 362–364

GD&T: Application and Interpretation

GD&T: Application and Interpretation, based on the ASME Y14.5-2009 *Dimensioning and Tolerancing* standard, is targeted to programs that require a study of geometric dimensioning and tolerancing as related to design, manufacturing, or inspection. This highly illustrated text contains topics ranging from the fundamentals of dimensioning to the extended principles of tolerance application and interpretation. Tolerance application and interpretation explanations are included for all of the categories of tolerances in the ASME Y14.5 standard. **GD&T: Application & Interpretation** covers practical applications of GD&T and the benefits of using GD&T in drawing documentation.

Illustrations are used extensively to clarify explanations. Color is used in illustrations to separate explanation data from the main portion of the figures. Color is also used to highlight instructional information such as tolerance placement requirements and tolerance zone boundaries.

Features...

- ✓ Provides in-depth explanations of the practices presented in the ASME Y14.5 standard.
- ✓ Includes hundreds of figures to illustrate ASME Y14.5 practices.
- ✓ Pro Tip, Note, Standards Advisory, and History Brief features highlight critical, difficult-to-understand, and historical topics.
- ✓ Review questions at the ends of chapters reinforce key concepts.
- ✓ Application problems at the ends of chapters provide activities for making tolerance calculations, dimensioning drawings, and adding tolerancing requirements to drawings.

About the Author...

Bruce A. Wilson has an international reputation as a leader in Dimensional Management and Geometric Dimensioning and Tolerancing. He is certified by the American Society of Mechanical Engineers (ASME) as a Senior Level Geometric Dimensioning and Tolerancing Professional. He has more than 25 years participation in ASME subcommittee Y14.5 standards development. He served as the leader of the positional tolerances section of the standard for more than 14 years and now is chairman of the Y14.5 subcommittee.

Mr. Wilson has extensive training and consulting experience. He has taught GD&T in Europe and the United States. His training experience includes curriculum development, teaching, management of training programs, and instructor development. Mr. Wilson's industrial experience includes responsibilities ranging from design engineer to program manager and industrial consultant. He was recognized as a Technical Fellow in The Boeing Company.

Mr. Wilson has contributed to the development of national and international standards since 1986 through his participation and membership on the ASME Y14.5 subcommittee. He has served as a member of the ASME Y14, Y14.1, Y14.2, Y14.3, and Y14.5 subcommittees and chaired both the Y14.3 and Y14.5 subcommittees.

Other Goodheart-Willcox titles...

- **CNC Machining**, by Richard A. Gizelbach
- **Drafting & Design**, by Clois E. Kicklighter and Walter C. Brown
- **Machining Fundamentals**, by John R. Walker and Bob Dixon
- **Print Reading for Industry**, by Walter C. Brown and Ryan K. Brown

For more information about Goodheart-Willcox titles, visit w

